FeView

FINITE ELEMENT MODEL (FEM) VISUALIZATION AND POST-PROCESSING TOOL FOR OPENSEES

Verification Report

PREPARED BY

MD MOTIUR RAHMAN, TAHMINA TASNIM NAHAR AND DOOKIE KIM

MATCH 2021

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

Contents

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

1. Verification (Linear Static Analysis):

1.1 Two Dimensional Problem:

1.1.1 Plane Frame:

1.1.1.1 Verification Example 1:

A cantilever concrete beam is subjected to a vertical load at the free end. The resulting vertical displacement measured at the free end of the beam obtained from the analysis program FeView is compared with SAP2000 results.

+ Geometry, Properties and Loading

Geometry and Loading:

10 kN

0.3m

5m

0.2m
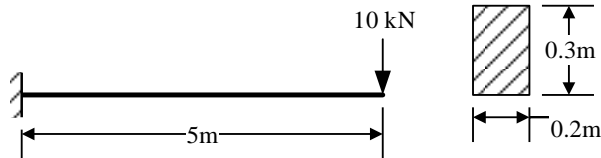
Material Properties:

$E = 30$ GPa
$v = 0.2$

+ Results and Comparison

The most significant results are compared in the table below:

| Output Parameter	FeView (20 element)	SAP2000 (20 element)
$U_{y,(free\ end)}$ [m]	-0.03086	-0.03094
$R_{z,(free\ end)}$ [rad]	-0.00926	- 0.00926
$RM_{z,(support)}$ [kN-m]	50	50

+ Comparison FeView & SAP2000 deform shape:

Shape Type	FeView	SAP2000
Deform shape		

+ Download TCL file:

https://www.dropbox.com/s/6of2h6nvqus9mfq/Example_1.tcl?dl=0

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

1.1.1.2 Verification Example 2:
A three-element frame is subjected to five load cases with various load types (point and distributed). Five different models have been created, one for each load case. The resulting displacements at specified joints obtained from analysis program FeView are compared with SAP200 results.
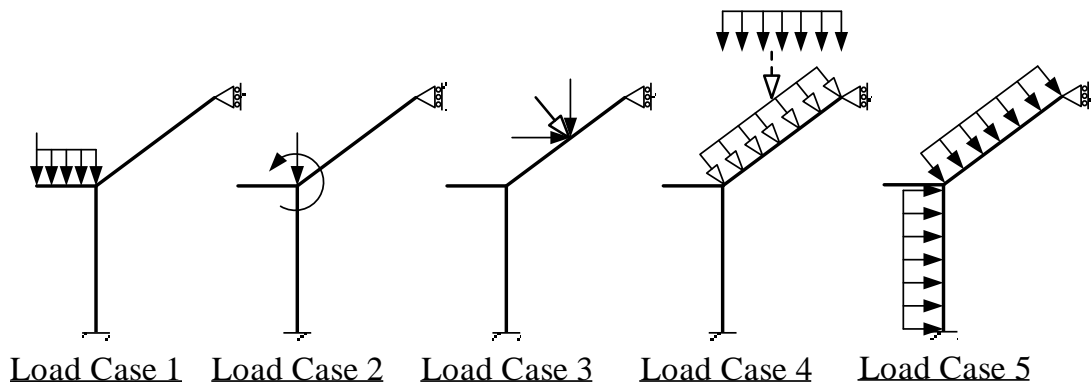
+ Geometry, Properties and Loading

Geometry:
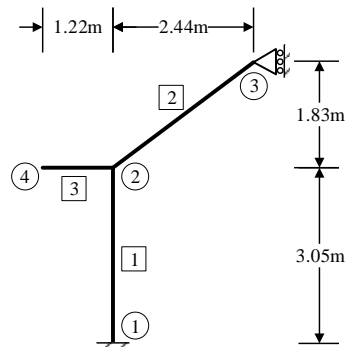
Material Properties:
E = 24.81989 GPa

Section Properties:
A = 0.0929 m^2
I = 7.1925e-4 m^4

Load Case 1   Load Case 2   Load Case 3   Load Case 4   Load Case 5

Modeling and Loading
• **Load Case 1 :** Distributed gravity load on frame element 3 and concentrated load on joint 4 (the uniform distributed load is inserted as element load in the local y direction (F_y= -26.27 kN/m) as the concentrated load is applied as point force in the global Y direction (F_y = -44.48 kN)

• **Load Case 2** Global point force and point moment at joint 2 (force and moment are applied as permanent loads, in terms of forces (F_y = -76.51 kN) and moments (M_z = 73.75 kNm) in the global Y and R_Z directions, respectively)

• **Load Case 3 :** Concentrated load on frame element 2 (it has been decomposed in a vertical and a horizontal component, so they are applied as permanent loads in terms of forces in the X and Y directions respectively, F_x = 40.032 kN and F_z = -53.376 kN)

• **Load Case 4 :** Distributed gravity load (global Y axis) 23.249 kN/m on frame element 2, applied as distributed load on the element local x and y axes, F_x = -23.349·cosφ = -14.009 kN/m and F_y = -23.349·sinφ =-18.679 kN/m, respectively. Element angle φ = arctan(1.83/2.44) = 36.9°

• **Load Case 5 :** Distributed horizontal load 29.186 kN/m on frame element 1 (global X axis), applied on the element local y axis. Distributed horizontal load -29.186 kN/m applied on frame element 2 local y axis.
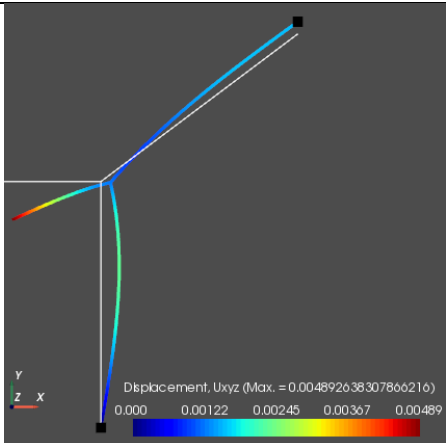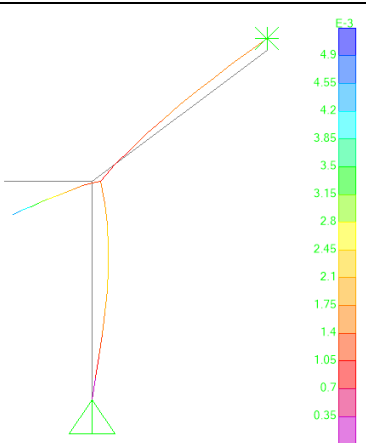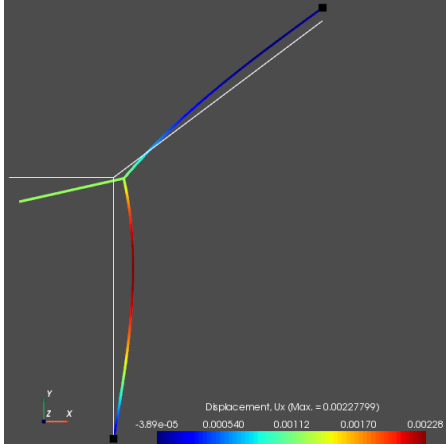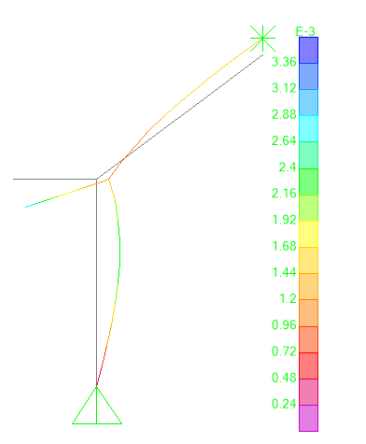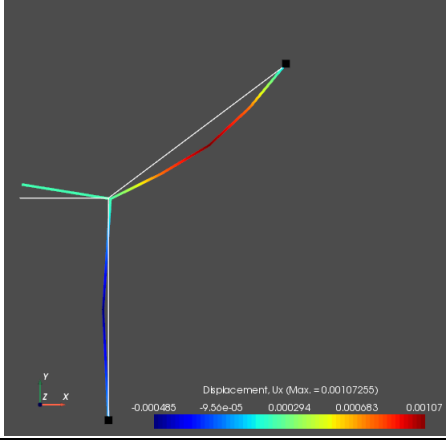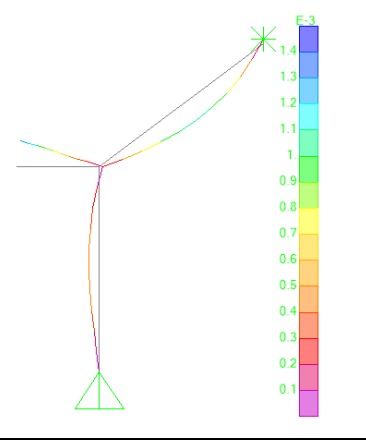
+ Results and Comparison

The most significant results are compared in the table below:

Load Case	Output Parameter	FeView	SAP2000
Case 1	$U_{y,(node\ 3)}$ [m]	0.001518	0.001599
Case 2	$U_{y,(node\ 3)}$ [m]	0.001517	0.001599
Case 3	$U_{x,(node\ 2)}$ [m]	0.000189	0.000165
Case 4	$U_{y,(node\ 3)}$ [m]	-0.007686	-0.007526

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

| Case 5 | $U_{x,(node\ 2)}$ [m] | 0.008005 | 0.007938 |

+ Comparison FeView & SAP2000 deform shape:

Load Case	FeView	SAP2000
Case 1		
Case 2		
Case 3		

5

Case 4		
Case 5		

+ Download TCL file:

Load Case 1: https://www.dropbox.com/s/hv61vhuon2kbbv1/Example_2_LoadCase1.tcl?dl=0
Load Case 2: https://www.dropbox.com/s/l0vopfcoif8r9v5/Example_2_LoadCase2.tcl?dl=0
Load Case 3: https://www.dropbox.com/s/rn6y0new7b9z2kq/Example_2_LoadCase3.tcl?dl=0
Load Case 4: https://www.dropbox.com/s/9ztk2xootkyoot3/Example_2_LoadCase4.tcl?dl=0
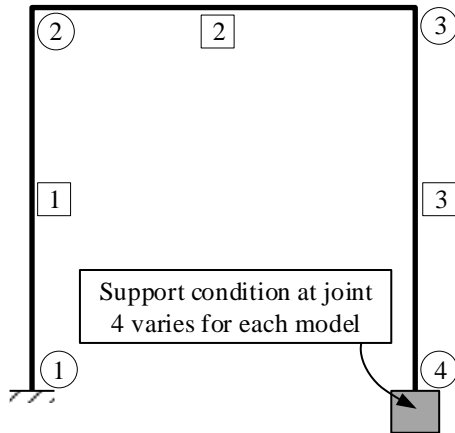Load Case 5: https://www.dropbox.com/s/ckfyj843mzv2q0s/Example_2_LoadCase5.tcl?dl=0

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

1.1.1.3 Verification Example 3:

This example tests FeView for settlement and rotation of normal supports and spring supports on a portal frame. Two different models have been created. The models are identical, except for the loading and the support condition at joint 4. The results obtained with the FE analysis program FeView at specified joints and in each model are compared with SAP2000 results.

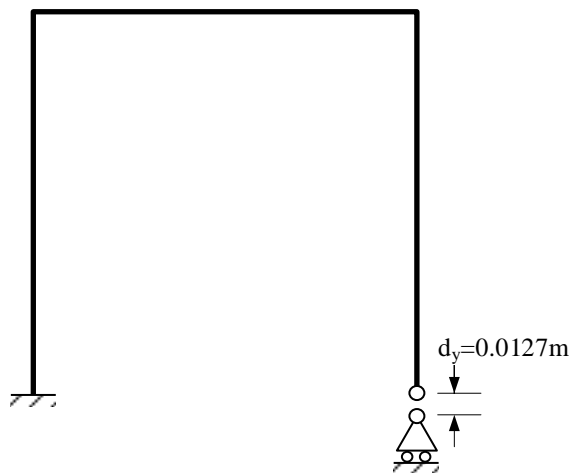- Geometry, Properties and Model Configuration:

Geometry:

Material Properties:
$E = 199.938$ GPa

Section Properties:
$A = 0.0929$ m^2
$I = 7.19248e\text{-}4$ m^4

Support condition at joint 4 varies for each model

$d_y = 0.0127$m

$r_z = 0.01$ rad
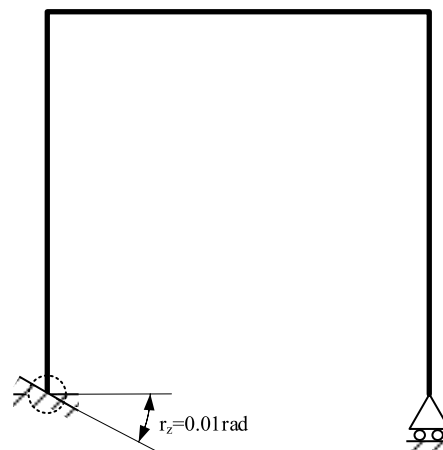
Model A

Model B

Results and Comparison

The most significant results are compared in the table below:

Model	Output Parameter	FeView	SAP2000
A	$RF_{y,(node\ 1)}$ [kN]	27.96	27.994
	$RM_{z,(node\ 1)}$ [kN-m]	-102.29	-102.393
B	$RF_{y,(node\ 1)}$ [kN]	-80.533	-80.624
	$RM_{z,(node\ 1)}$ [kN-m]	294.59	294.890

+ Comparison FeView & SAP2000 deform shape:

Model	FeView	SAP2000

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

A		
B		
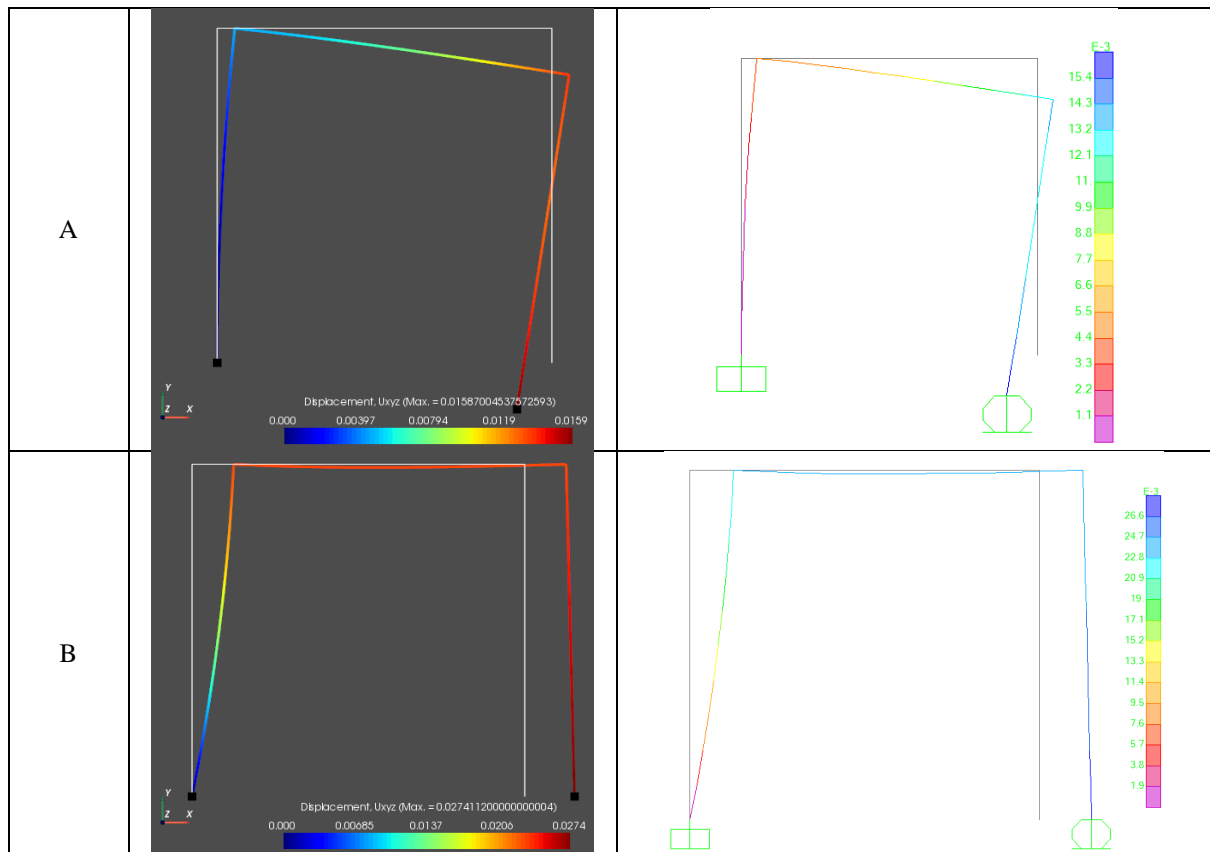
+ Download TCL file:

Model A: https://www.dropbox.com/s/6iiv7r22rqswmwf/Example_3A.tcl?dl=0
Model B: https://www.dropbox.com/s/r23lsrpfhc39u3y/Example_3B.tcl?dl=0

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

1.1.2 Plane Truss:

1.1.2.1 Verification Example 4:

A steel truss is subjected to is subjected to vertical loads as shown in figure below. The resulting maximum vertical displacement obtained from the analysis program FeView is compared with ABAQUS results.

+ Geometry, Properties and Loading

Geometry and Loading:

10 kN   10 kN   10 kN   10 kN   10 kN   10 kN   10 kN

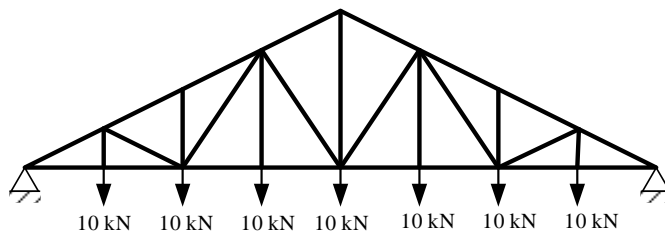**Material Properties:**
E = 200 GPa
v = 0.2

Section Properties:
A = 0.002 m^2

+ Results and Comparison

The most significant results are compared in the table below:

Output Parameter	FeView	ABAQUS
$U_{y,(max)}$ [m]	-0.00181	-0.00181

+ Comparison FeView & SAP2000 deform shape:

Software	Deform shape
FeView	
ABAQUS	

+ Download TCL file:
https://www.dropbox.com/s/08kx10068ozj9m2/Example_4.tcl?dl=0

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

1.1.3 Planes:

1.1.3.1 Verification Example 5:

In this example, a straight cantilever beam, modelled with plane stress elements, is subjected to forces at the tip in the X direction .The tip displacements in the X direction obtained from the analysis program FeView is compared with SAP2000 results.

+ Geometry, Properties and Loading

Geometry and Loading:
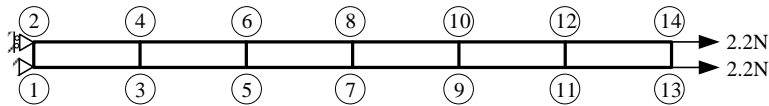
Material Properties:

E = 68.95 GPa

v = 0.3

G = 26.52 GPa

Section Properties:
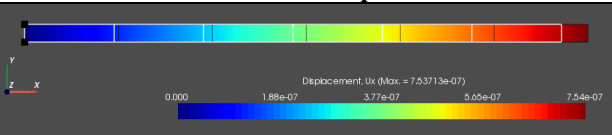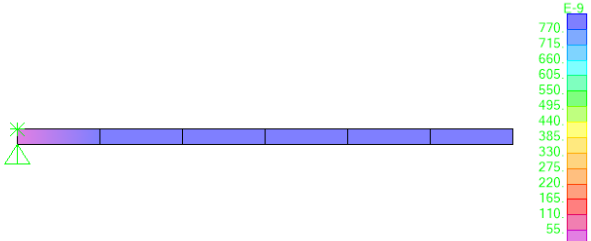
Plane element thickness = 2.54×10^{-3} m

+ Results and Comparison

The most significant results are compared in the table below:

Output Parameter	FeView	SAP2000
$U_{x,(node\ 13)}$ [m]	0.00000075	0.00000076

+ Comparison FeView & SAP2000 deform shape:

Software	Deform shape
FeView	
SAP2000	

+ Download TCL file:

https://www.dropbox.com/s/7ges1lsvvi4m5tq/Example_5.tcl?dl=0

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

1.1.3.2 Verification Example 6:

In this example, a rectangular plate with irregularly shaped elements is subjected to prescribed displacements at the edges that theoretically impose a constant stress field over the model. The geometry, properties and loading are as described in MacNeal and Harder 1985. The plane stress element is used in FeView and the membrane stress components resulting from the prescribed displacements are compared with SAP2000.

The Ux and Uy degrees of freedom are active for the analysis. All other degrees of freedom are inactive. Joints 1, 4, 7 and 8 are restrained for translation in the X and Y directions. The prescribed displacements are applied to the restrained degrees of freedom of those joints.

The plane section is modeled using the plane stress element.

+ Geometry, Properties and

Geometry and Coordinates:

Joint	X (m)	Y (m)
1	0.0061	0.00305
2	0.00406	0.00203
3	0.00457	0.00076
4	0.0061	0.0
5	0.00203	0.00203
6	0.00102	0.00051
7	0.0	0.00305
8	0.0	0.0

Material Properties:
E = 68.95 GPa
v = 0.25

Section Properties:
Plane element thickness = 2.54×10^{-3} m
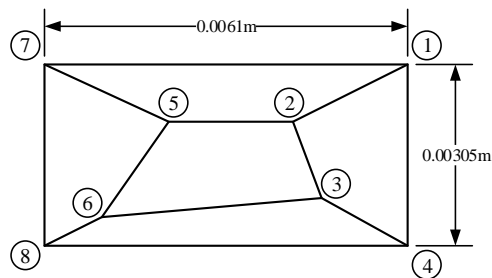
+ Loading

The loading is provided in the form of prescribed edge displacements Ux and Uy, which are imposed on joints 1, 4, 7 and 8. Those displacements are defined by the following equations.
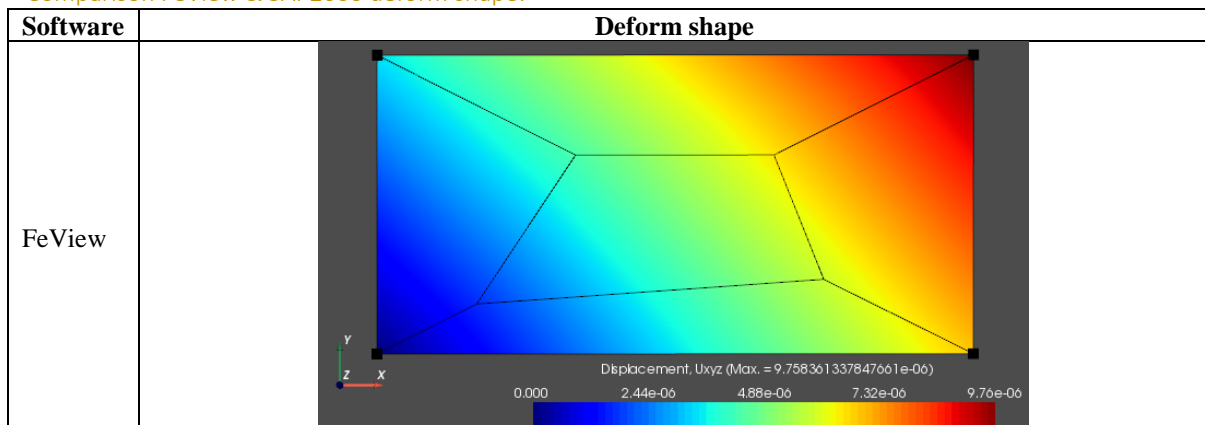
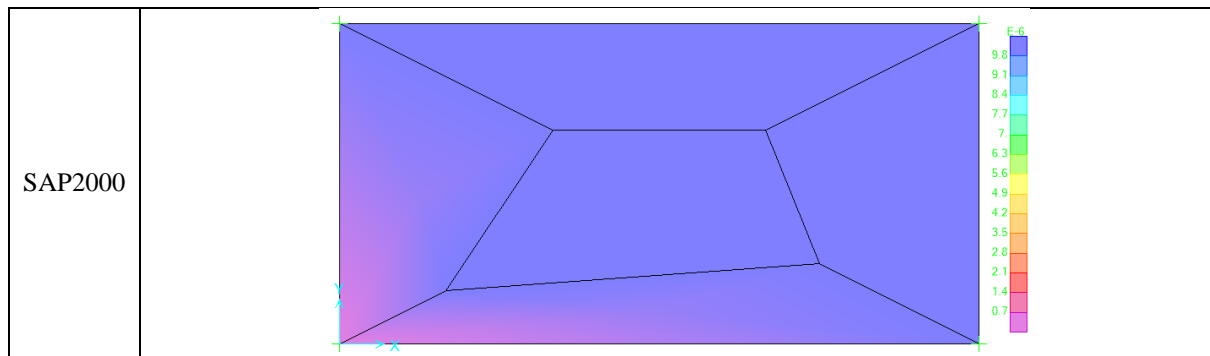$$U_x = \frac{x + \frac{y}{2}}{1000}, \qquad U_y = \frac{y + \frac{x}{2}}{1000}$$

+ Results and Comparison

The most significant results are compared in the table below:

Joint	FeView		SAP2000	
	Ux	Uy	Ux	Uy
2	5.07e-06	4.06e-06	5.08E-06	4.06E-06
3	4.95e-06	3.04e-06	4.95E-06	3.04E-06
5	3.04e-06	3.04e-06	3.04E-06	3.04E-06
6	1.27e-06	1.02e-06	1.27E-06	1.02E-06

+ Comparison FeView & SAP2000 deform shape:

Software	Deform shape
FeView	

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

SAP2000		

+ Download TCL file:
https://www.dropbox.com/s/xp6ioyypotu9c1h/Example_6.tcl?dl=0

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

1.2 Three Dimensional Problem:
1.2.1 Space Frame:

1.2.1.1 Verification Example 7:

A space frame is subjected to load on X and Y direction as shown in figure. The displacements (Ux, Uy, Uz) measured at joint 1 end of the beam are obtained from the analysis program FeView is compared with SAP2000 results.

+ Geometry, Properties and Loading

Geometry and Loading:

Material Properties:

E = 30 GPa

v = 0.2
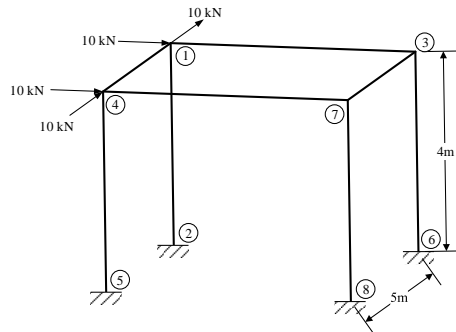
Section Properties:

I $= 1.33 \times 10^4$ m

+ Results and Comparison

The most significant results are compared in the table below:

Output Parameter	FeView	SAP2000
$U_{x,(node\ 1)}$ [m]	0.0125375	0.012612
$U_{y,(node\ 1)}$ [m]	0.0164007	0.016496
$U_{z,(node\ 1)}$ [m]	-3.74e-06	-3.73E-06

+ Comparison FeView & SAP2000 deform shape:

Shape type	FeView	SAP2000
Deform shape		

+ Download TCL file:

https://www.dropbox.com/s/it08gq0ps1zf5di/Example_7.tcl?dl=0

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

1.2.2 Solid:

1.2.2.1 Verification Example 8:

A Solid simple beam is subjected to load on X and Y direction as shown in figure. The displacements (Ux, Uy, Uz) measured at joint 1 end of the beam are obtained from the analysis program FeView is compared with SAP2000 results.

+ Geometry, Properties and Loading

Geometry and Loading:

1320 kN/m²

0.5m

0.5m

5m

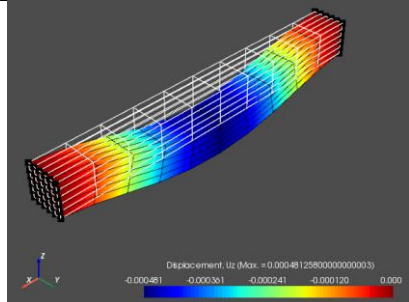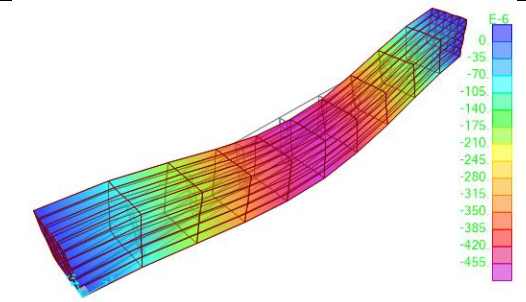**Material Properties:**

E = 30 GPa

v = 0.2
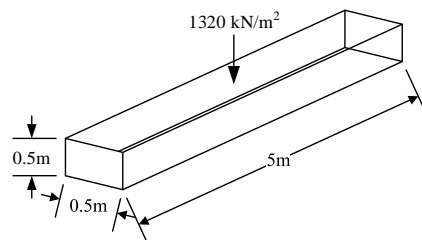
+ Results and Comparison

The most significant results are compared in the table below:

Output Parameter	FeView (250 brick element)	SAP2000 (250 brick element)
$U_{z,(max)}$ [m]	-0.000481	-0.000481
$U_{y,(node\ 1)}$ [m]	0.0164007	0.016496
$U_{z,(node\ 1)}$ [m]	-3.74E-06	-3.73E-06

+ Comparison FeView & SAP2000 deform shape:

Shape type	FeView (250 brick element)	SAP2000 (250 brick element)
Deform shape		

+ Download TCL file:

https://www.dropbox.com/s/lw1m9xitclbvar2/Example_8.tcl?dl=0

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php
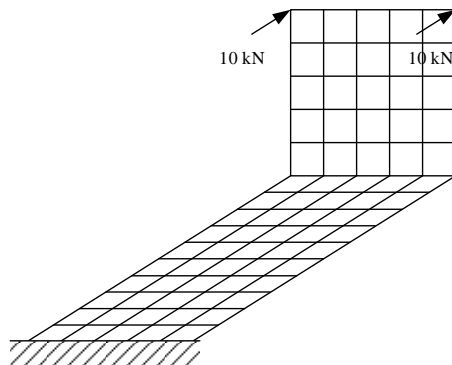
1.2.3 Shell:

1.2.3.1 Example 9:
A steel angle modled by shell element is loaded as shown in figure. The maximum displacements is X and Z direction is measured at the loaded point of the angle are obtained from the analysis program FeView is compared with SAP2000 results.

+ Geometry, Properties and Loading

Geometry and Loading:

Material Properties:
E = 200 GPa
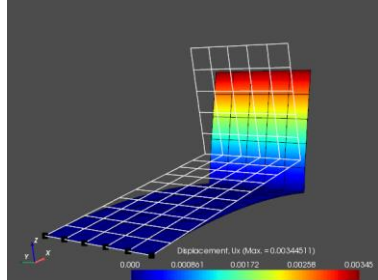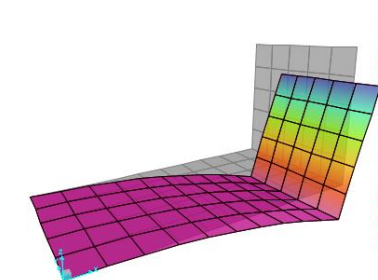v = 0.3
Section Properties:
Shell thickness = 0.02 m

+ Results and Comparison

The most significant results are compared in the table below:

Output Parameter	FeView	SAP2000
$U_{x,(max)}$ [m]	0.0034	0.0034
$U_{z,(max1)}$ [m]	-0.0029	-0.0029

+ Comparison FeView & SAP2000 deform shape:

Shape type	FeView	SAP2000
Deform shape		

+ Download TCL file:
https://www.dropbox.com/s/9v1rr8qqt1aj585/Example_9.tcl?dl=0

1.2.4 Space Truss:

1.2.4.1 Verification Example 10:
A steel space truss is loaded in X and Z direction at joint 1.. The displacements is X and Z direction is measured at joint 1 of the truss obtained from the analysis program FeView is compared with SAP2000 results.

+ Geometry, Properties and Loading

Geometry and Loading:
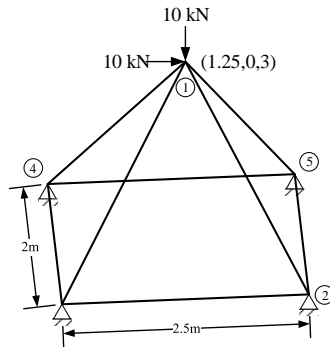
Material Properties:
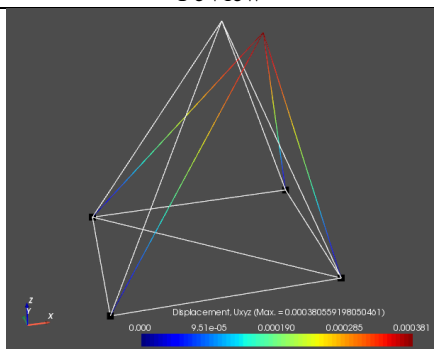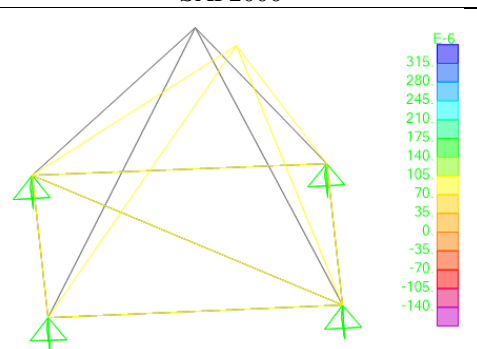E = 200 GPa

Section Properties:
A = 0.001 m2

+ Results and Comparison

The most significant results are compared in the table below:

Output Parameter	FeView	SAP2000
$U_{x,(node\ 1)}$ [m]	3.40E-4	3.40E-4
$U_{z,(node\ 1)}$ [m]	-9.54E-5	-9.54e-5

+ Comparison FeView & SAP2000 deform shape:

Shape type	FeView	SAP2000
Deform shape		

+ Download TCL file:
https://www.dropbox.com/s/jla5tp5yjpxm8j6/Example_10.tcl?dl=0

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

2. Verification (Modal Analysis):

## 2.1 Two Dimensional Problem:

2.1.1 Plane Frame:

2.1.1.1 Verification Example 11:

A ten bay, nine story, two dimensional frame structure solved in Bathe and Wilson (1972) is analyzed for the first three eigenvalues. The material and section properties, and the mass per unit length used for all members, as shown below, are consistent with those used in the above-mentioned reference.

The results obtained with the FE analysis program FeView are compared with SAP2000 results.

+ Geometry, Properties

Geometry:
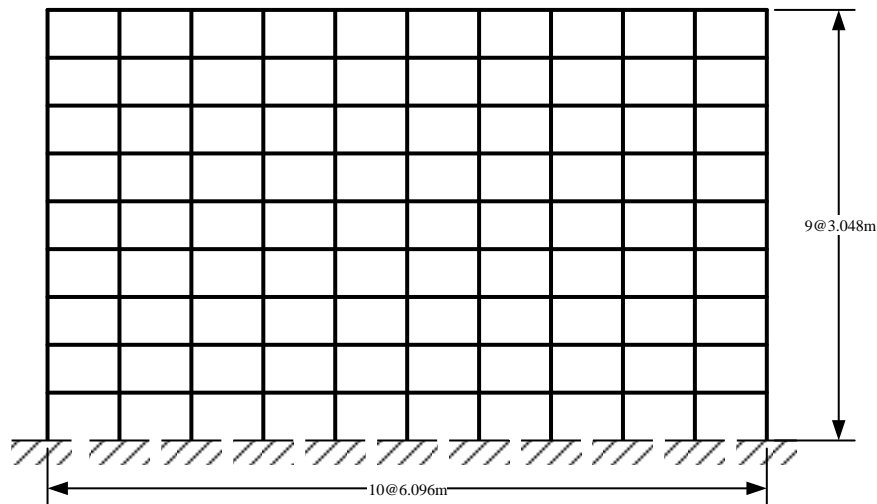
Material Properties:
E = 20.6832410 GPa
Section Properties:
A = 0.279 m2
I = 0.008631 m4

9@3.048m

10@6.096m
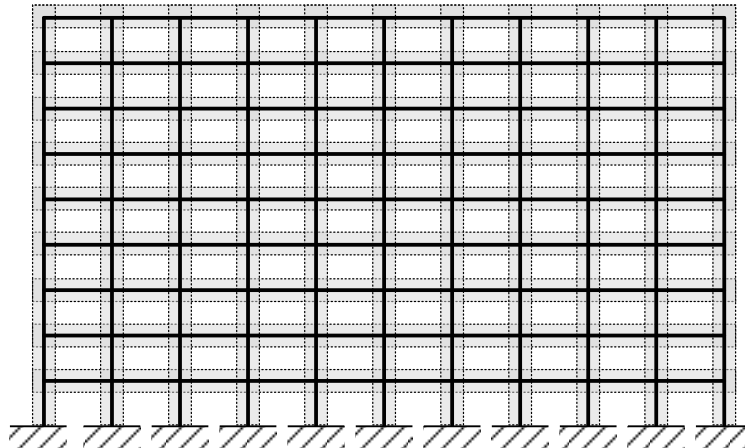
+ Loading/Modeling

The frame objects are modeled through elastic frame elements with specified Mass/Length^3 = 514.81 ton/m3. All the base nodes are fully restrained.
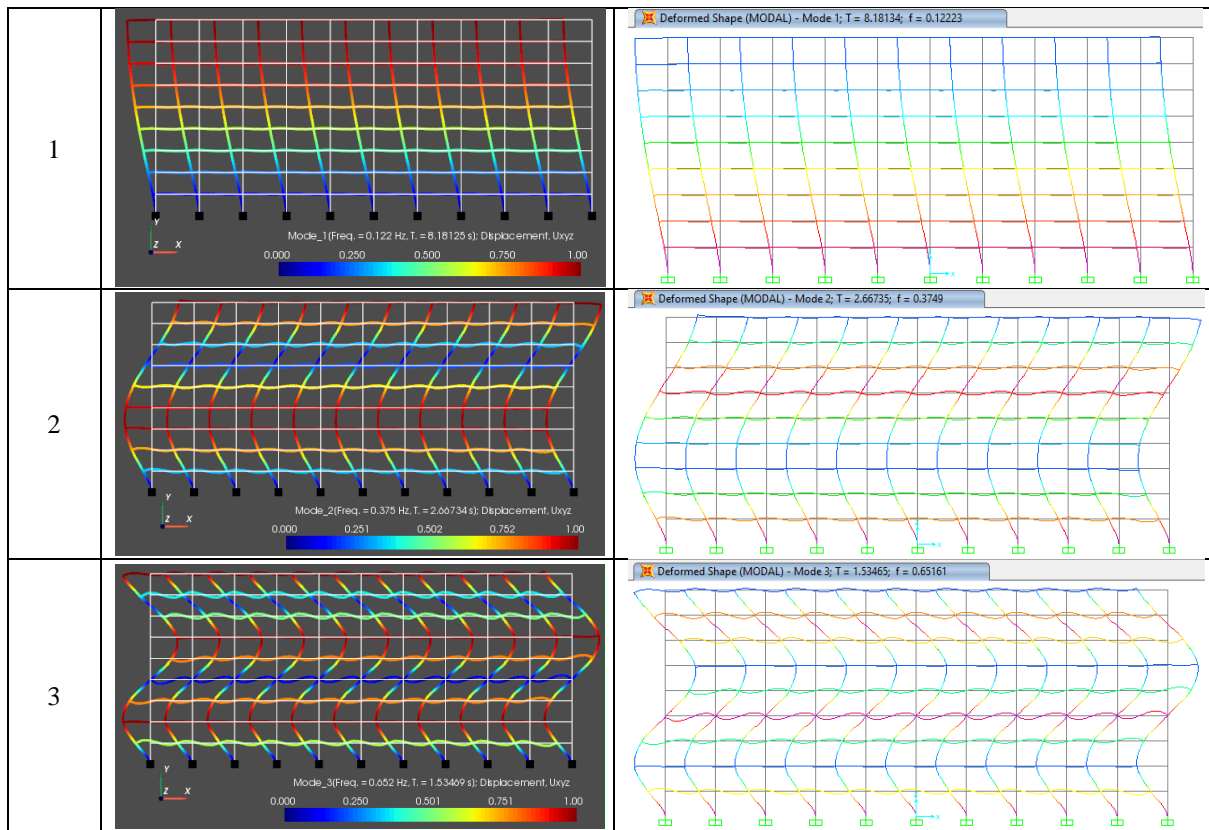
The FE model is presented below:

+ Results and Comparison

The most significant results are compared in the table below:

Output Parameter	GiD+OpenSees	SAP2000
Frequency F_1 [Hz] (1st mode)	0.122	0.122
Frequency T_1 [Hz] (2nd mode)	0.375	0.375
Frequency T_1 [Hz] (3rd mode)	0.652	0.0.652

+ Comparison FeView & SAP2000 deform shape:

Mode	FeView	SAP2000

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

1		
2		
3		

+ Download TCL file:
https://www.dropbox.com/s/v09o6t13w8e7uw2/Example_11.tcl?dl=0

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

2.1.2 Plane Truss:
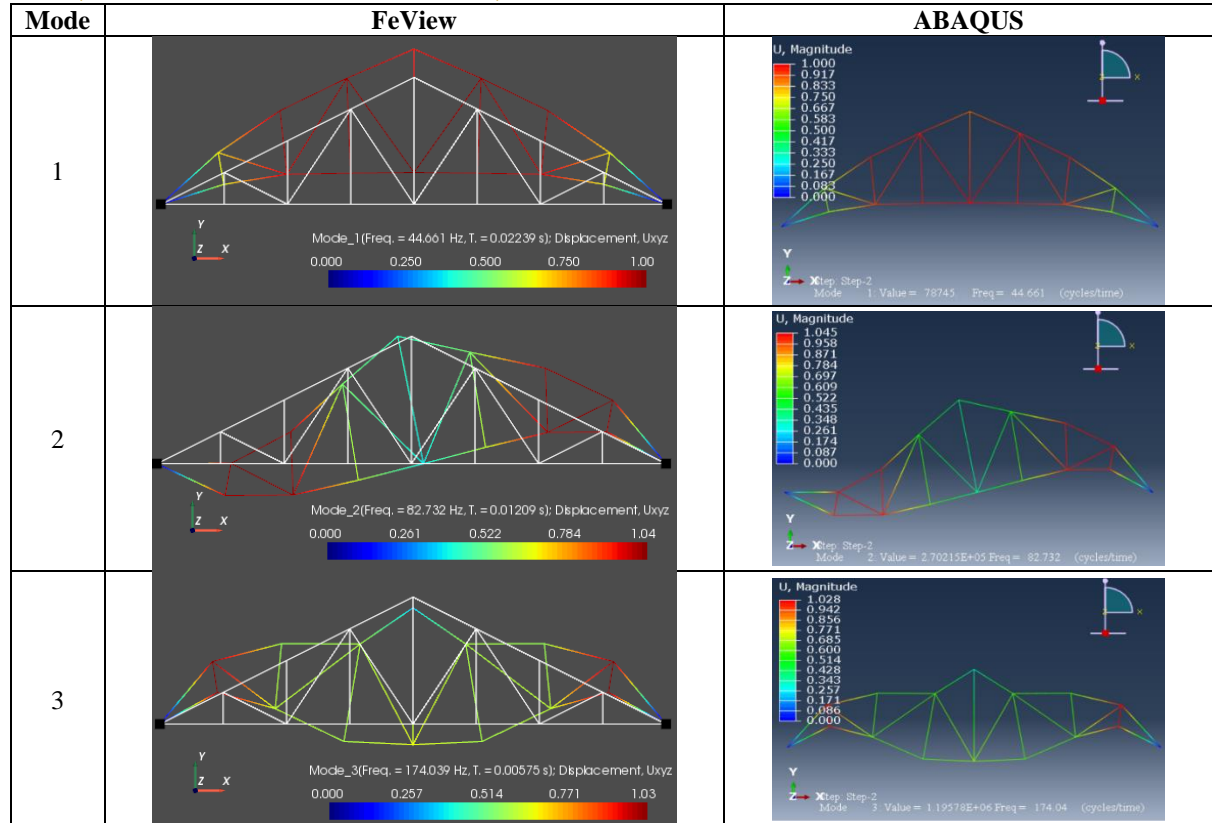
2.1.2.1 Verification Example 12:

Example 4 with material density 7850 kg/m3 is considered for this example analysed for the first three mode. The results obtained with the FE analysis program FeView are compared with ABAQUS results.

+ Results and Comparison

The most significant results are compared in the table below:

Output Parameter	FeView	ABAQUS
Frequency F_1 [Hz] (1st mode)	44.661	44.661
Frequency F_2 [Hz] (2nd mode)	82.732	82.732
Frequency F_3 [Hz] (3rd mode)	174.04	174.04

+ Comparison FeView & SAP2000 deform shape:

Mode	FeView	ABAQUS
1		
2		
3		

+ Download TCL file:

https://www.dropbox.com/s/9zbd32k4zn7qgie/Example_12.tcl?dl=0

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php
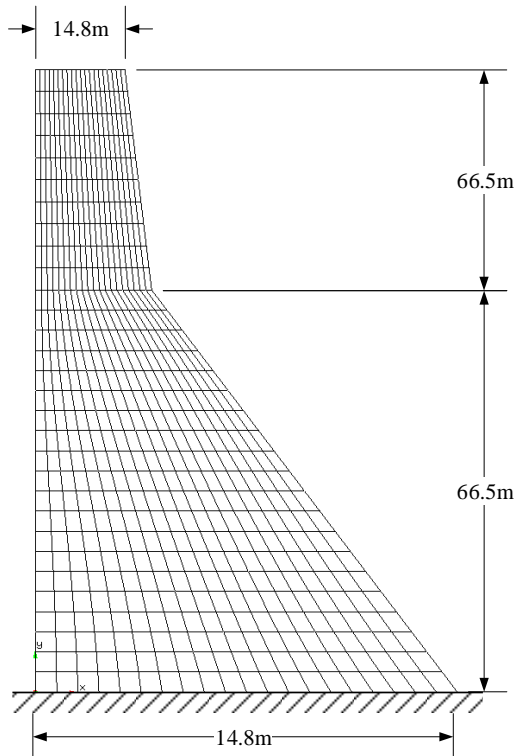
2.1.3 Plane:

2.1.3.1 Verification Example 13:
In this example we consider an analysis of the Koyna dam which is modelled by plain strain element and analysed for the first three mode.
The results obtained with the FE analysis program FeView are compared with SAP2000 results.
+ Geometry, Properties

Geometry:

14.8m

66.5m

66.5m

14.8m

Material Properties:
E = 31.027 GPa
ν = 0.15
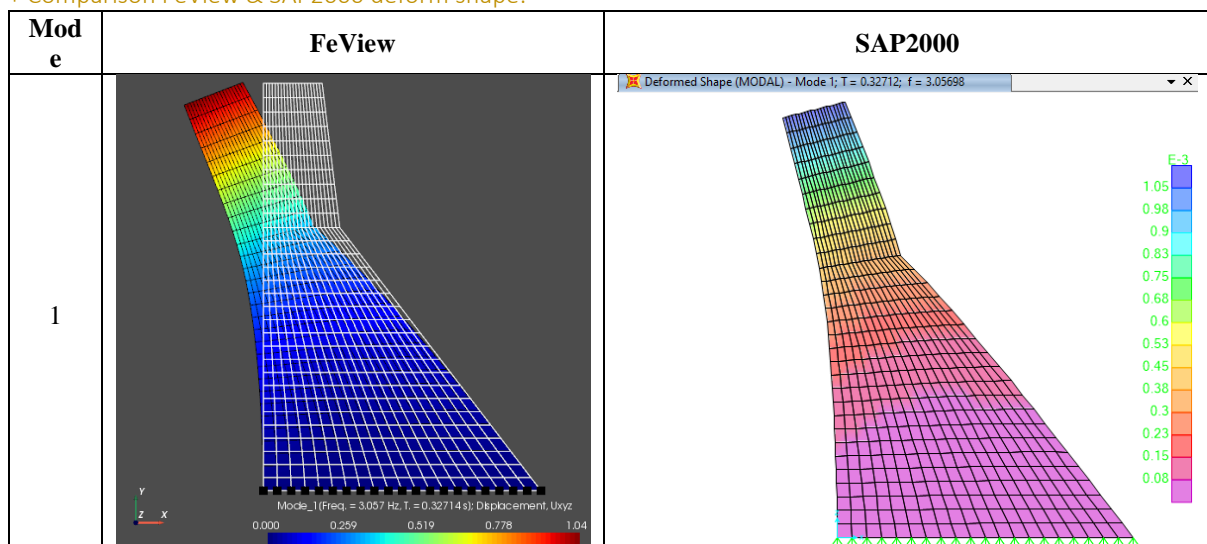ρ = 2643 kg/m3

Section Properties:
Plane element thickness = 1 m

+ Results and Comparison
The most significant results are compared in the table below:

Output Parameter	FeView	SAP2000
Frequency F1 [Hz] (1st mode)	3.057	3.057
Frequency F2 [Hz] (2nd mode)	8.133	8.134
Frequency F3 [Hz] (3rd mode)	10.977	0.0.652

+ Comparison FeView & SAP2000 deform shape:

Mode	FeView	SAP2000
1		

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

2		
3		
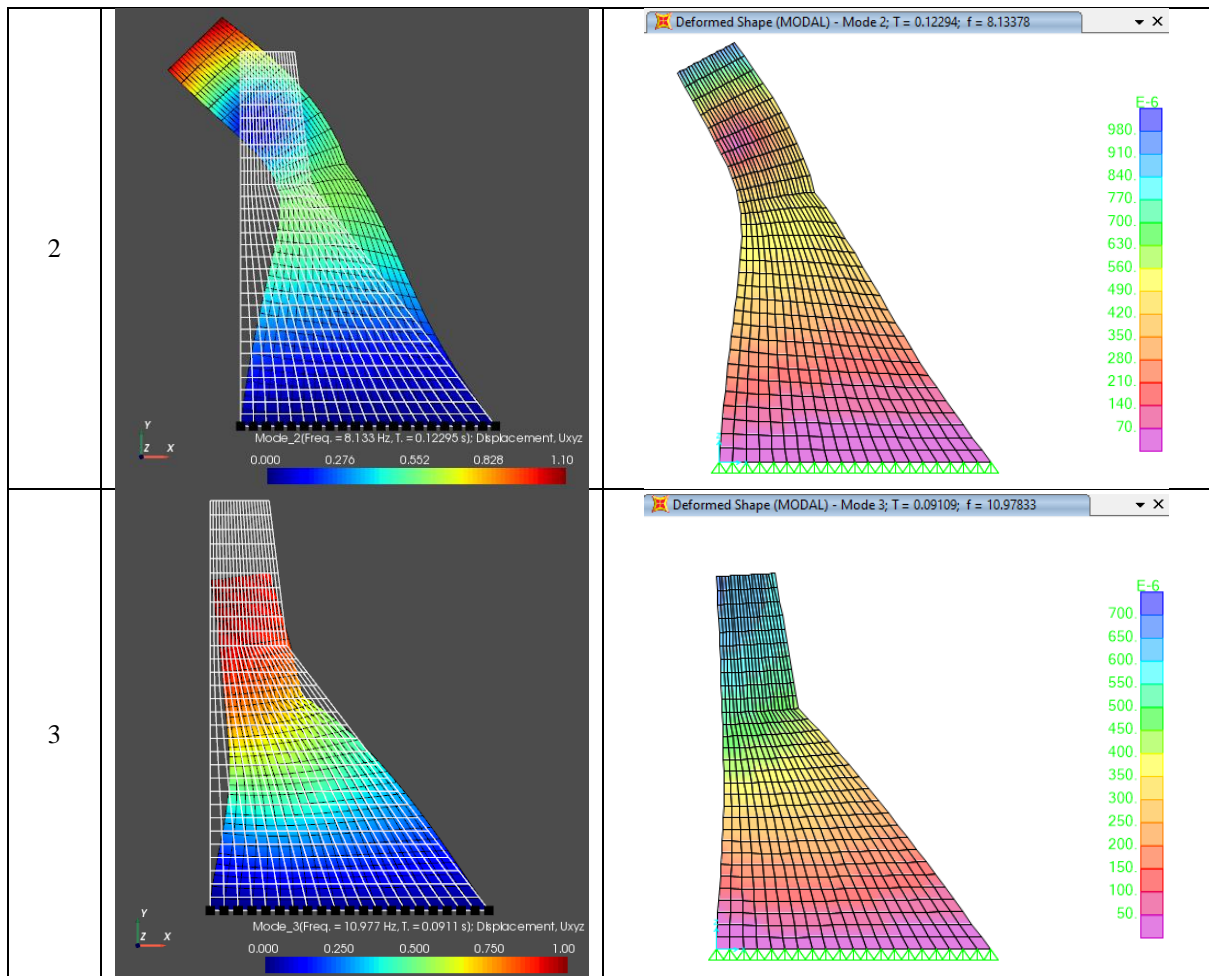
+ Download TCL file:
https://www.dropbox.com/s/s7jnuychejwfbs0/Example_13.tcl?dl=0

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php
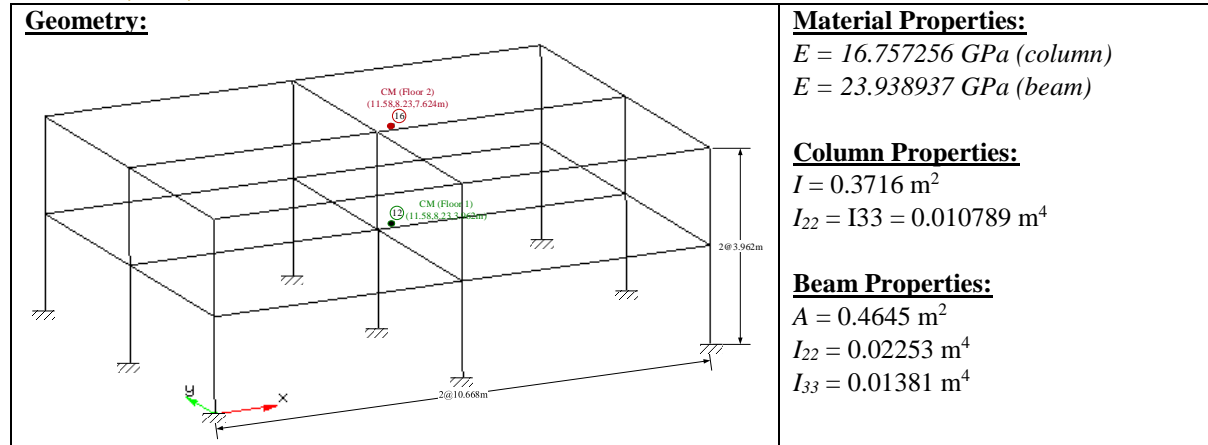
2.2 Three Dimensional Problem:
2.2.1 Space Frame:

2.2.1.1 Verification Example 14:

A two-story, two- bay, three-dimensional frame structure is analyzed for its four natural frequencies. The structure is doubly symmetric in plan, except that the center of mass at each story level is eccentric, as shown in the figure below. The entire story mass is applied at these joints in the X and Y directions only.

The FeView results are compared with SAP2000 results.

+ Geometry, Properties

Geometry:	Material Properties:
	E = 16.757256 GPa (column) *E = 23.938937 GPa (beam)* **Column Properties:** *I = 0.3716 m²* *I_22 = I33 = 0.010789 m⁴* **Beam Properties:** *A = 0.4645 m²* *I_22 = 0.02253 m⁴* *I_33 = 0.01381 m⁴*

+ Loading/Modeling

A lumped mass is applied to joints 28 and 29 with a value of 90.64566 ton in the X and Y directions. Two rigid diaphragm constraints are introduced (one at each floor level). All the base nodes are fully restrained.
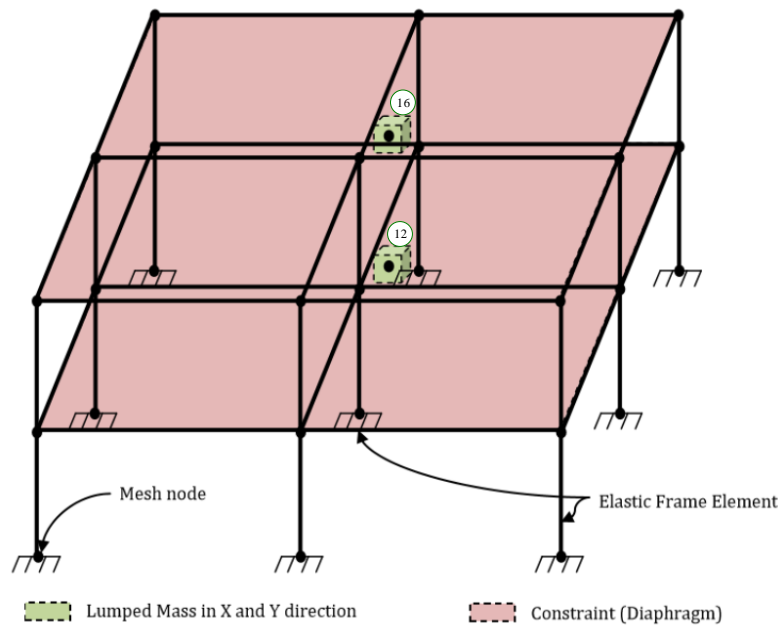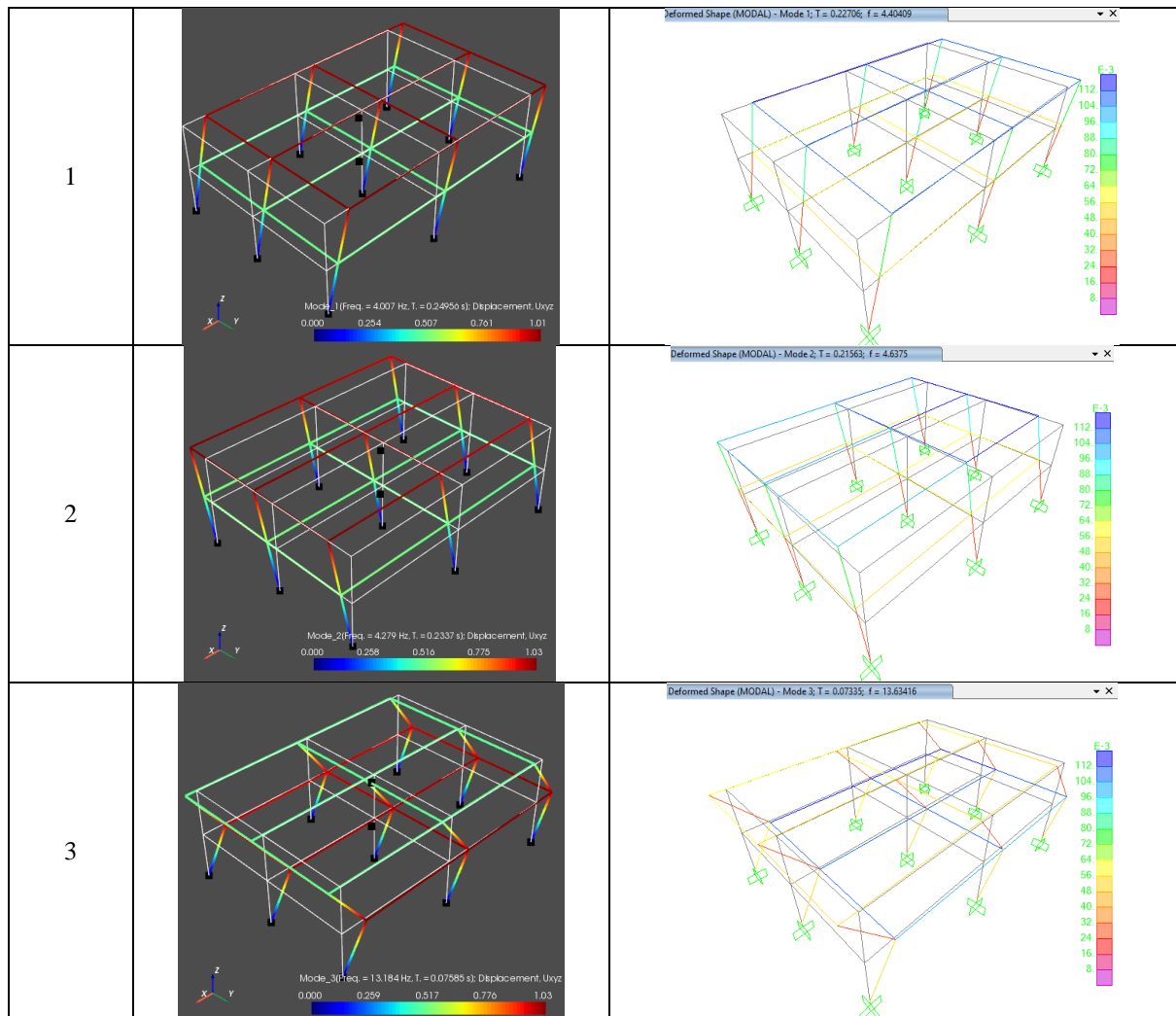
The FE model is presented below:

+ Results and Comparison

The most significant results are compared in the table below:

Output Parameter	Output Parameter	FeView	SAP2000
Frequency F1 [Hz] (1st mode)	$U_{x,(node\ 1)}$ [m]	0.0125375	0.012612
Frequency F2 [Hz] (2nd mode)	$U_{y,(node\ 1)}$ [m]	0.0164007	0.016496
Frequency F3 [Hz] (3rd mode)	$U_{z,(node\ 1)}$ [m]	-3.74e-06	-3.73E-06

+ Comparison FeView & SAP2000 deform shape:

Mode	FeView	SAP2000

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

1			
2			
3			

+ Download TCL file:
https://www.dropbox.com/s/9utmgm698dhpuhd/Example_14.tcl?dl=0

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

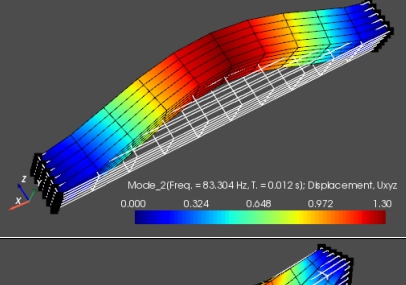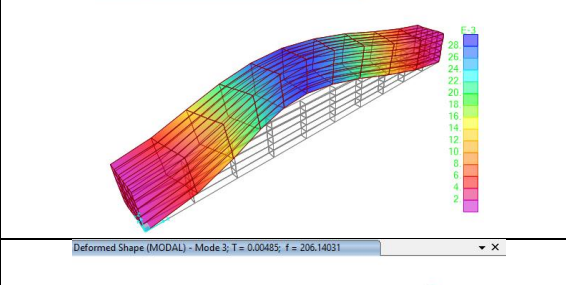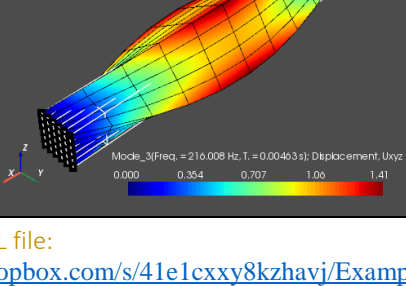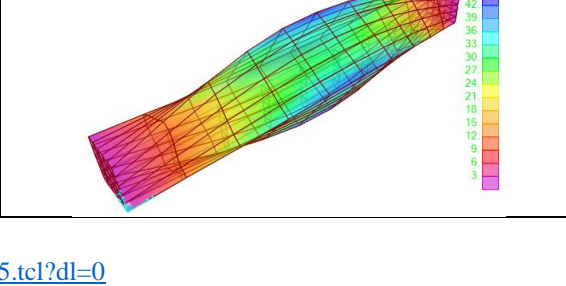### 2.2.2 Solid:

2.2.2.1 Verification Example 15:
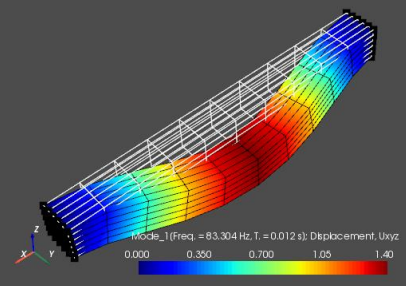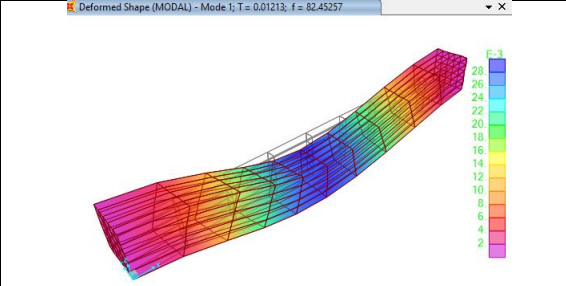
Example 8 with material density 2400 kg/m3 is considered for this example and analysed for the first three modes. The results obtained with the FE analysis program FeView are compared with SAP2000 results..

+ Results and Comparison

The most significant results are compared in the table below:

Output Parameter	FeView	SAP2000
Period T_1 [s] (1st mode)	0.0120	0.0121
Period T_2 [s] (2nd mode)	0.0120	0.0121
Period T_3 [s] (3rd mode)	0.0046	0.0049

+ Comparison FeView & SAP2000 deform shape:

Mode	FeView	SAP2000
1		
2		
3		

+ Download TCL file:

https://www.dropbox.com/s/41e1cxxy8kzhavj/Example_15.tcl?dl=0

2.2.3 Shell:

2.2.3.1 Verification Example 16:

Example 9 with material density 7850 kg/m3 is considered for this example and analysed for the first three modes. The results obtained with the FE analysis program FeView are compared with SAP2000 results..
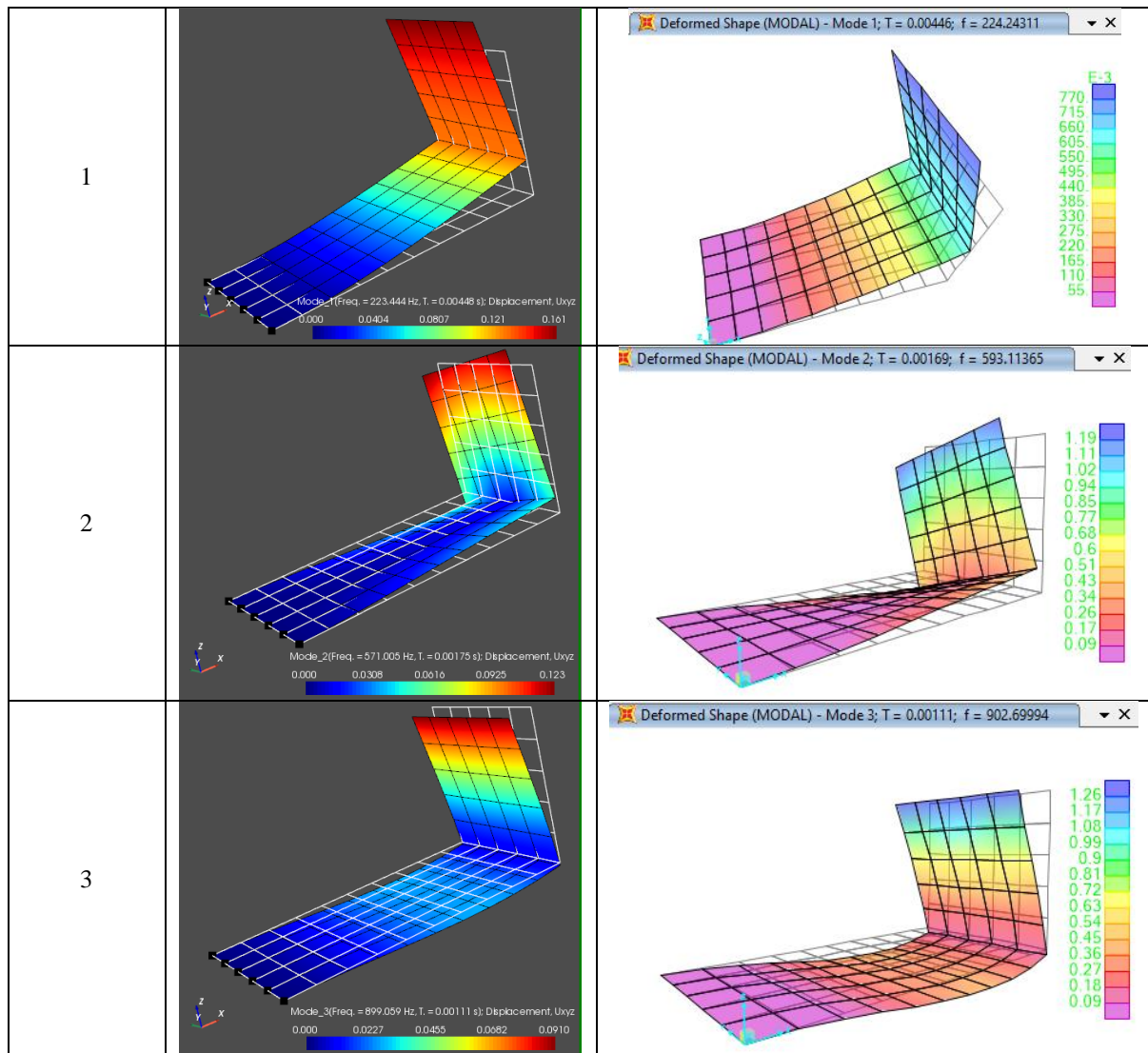
+ Results and Comparison

The most significant results are compared in the table below:

Output Parameter	FeView	SAP2000
Period T1 [s] (1st mode)	0.0045	0.0045
Period T2 [s] (2nd mode)	0.0018	0.0017
Period T3 [s] (3rd mode)	0.0011	0.0011

+ Comparison FeView & SAP2000 deform shape:

Mode	FeView	SAP2000

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

1		
2		
3		

+ Download TCL file:
https://www.dropbox.com/s/lcdhco2f4q6rn4l/Example_16.tcl?dl=0

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php

2.2.4 Space Truss:

2.2.4.1 Verification Example 17:
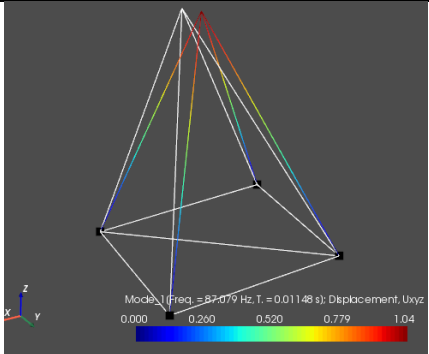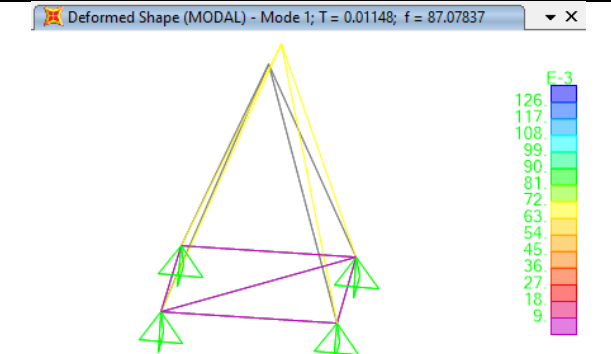Example 10 with material density 7850 kg/m3 is considered for this example and analysed for the first three modes.
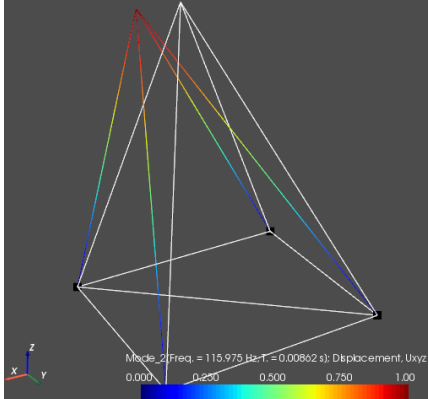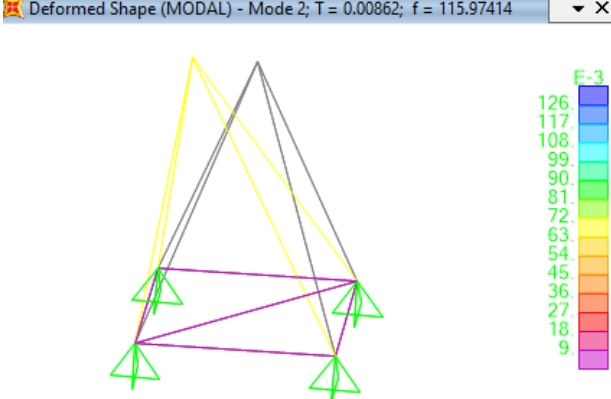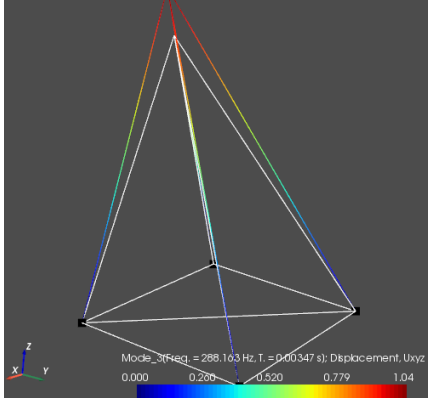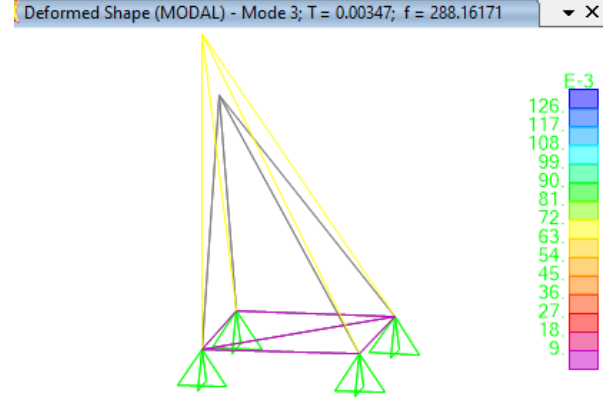The results obtained with the FE analysis program FeView are compared with SAP2000 results..

+ Results and Comparison

The most significant results are compared in the table below:

Output Parameter	FeView	SAP2000
Period T1 [s] (1st mode)	0.01148	0.01148
Period T2 [s] (2nd mode)	0.00862	0.00862
Period T3 [s] (3rd mode)	0.00347	0.00347

+ Comparison FeView & SAP2000 deform shape:

Mode	FeView	SAP2000
1		
2		
3		

+ Download TCL file:

https://www.dropbox.com/s/2etybjd3md1i4y6/Example_17.tcl?dl=0

For more details visit: http://www.kim2kie.com/3_ach/SSL_Software.php