

FeView

A TOOL FOR FINITE ELEMENT MODEL (OPENSEEES) VISUALIZATION

MANUAL

PREPARED BY

MD MOTIUR RAHMAN, TAHMIN TASNIM NAHAR, TRAN THANH TUAN AND DOOKIE KIM



MATCH 2020

Contents

Contents	2
1. Interactive Interface	4
2. Installation	5
3. Active Tcl (required)	6
3.1 Download:	6
3.2 Instalation of tcl/tk	6
4. Features.....	7
4.1 Analysis Type:	7
4.1.1 Static Analysis (linear/non linear)	7
4.1.2 Modal Analysis	7
4.1.3 Dynamic Analysis	7
4.2 Model Type:.....	7
4.3 Materials:	7
4.4 Elements	7
4.4.1 Truss Elements:.....	7
4.4.2 Beam-Column Elements:	7
4.4.3 Link Elements:	7
4.4.4 Bearing Elements:	7
4.4.5 Quadrilateral Elements:.....	7
4.4.6 Brick Elements:.....	8
4.4.7 Cable Elements	8
4.4.8 Tetrahedron Elements	8
4.4.9 Triangular Elements	8
5. Recorder Setting :	8
5.1 Node recorder	8
5.2 Element recorder.....	8
6. Modal Analysis Setting.....	8
6.1 Mode Number	8
6.2 Record eigenvectors.....	8
7. Verification (Linear Static Analysis):	9
7.1 Two Dimensional Problem:	9
7.1.1 Plane Frame:	9
7.1.2 Plane Truss:	15
7.1.3 Planes:	16
7.2 Three Dimensional Problem:	19
7.2.1 Space Frame:	19
7.2.2 Solid:	20
7.2.3 Shell:	21

7.2.4 Space Truss:	22
8. Verification (Modal Analysis):	23
8.1 Two Dimensional Problem:	23
8.1.1 Plane Frame:	23
8.1.2 Plane Truss:	25
8.1.3 Plane:	26
8.2 Three Dimensional Problem:	28
8.2.1 Space Frame:	28
8.2.2 Solid:	30
8.2.3 Shell:	30
8.2.4 Space Truss:	32
9. Example:	33
9.1 Static Pushover Analysis (Non-linear Analysis):	33
+ Geometry, Properties	33
+ Loading/Modeling (for making TCL file)	33
+ Open Model:	33
+ Run OpenSees and show deformed model:	33
+ Pushover Curve:	34
+ Download TCL file:	35
9.2 Dynamic Analysis:	36
+ Geometry, Properties	36
+ Mode Shape:	36
+ Deformed shape for transient analysis:	36
+ Response plot for transient analysis:	37
+ Response Spectrum:	38

1. Interactive Interface

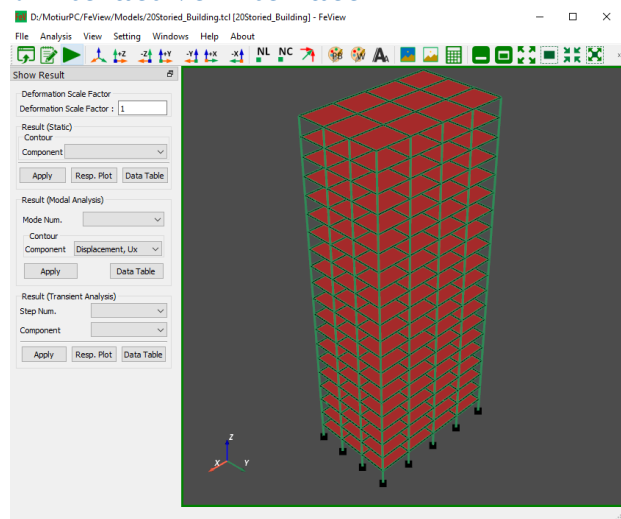


Figure: High rise building

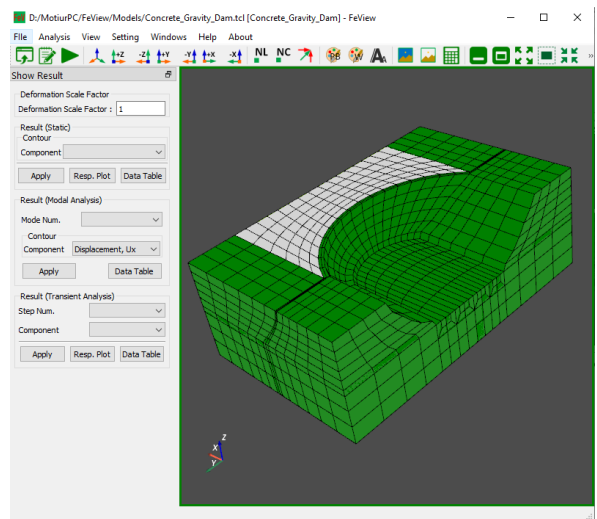


Figure: Concrete Gravity dam

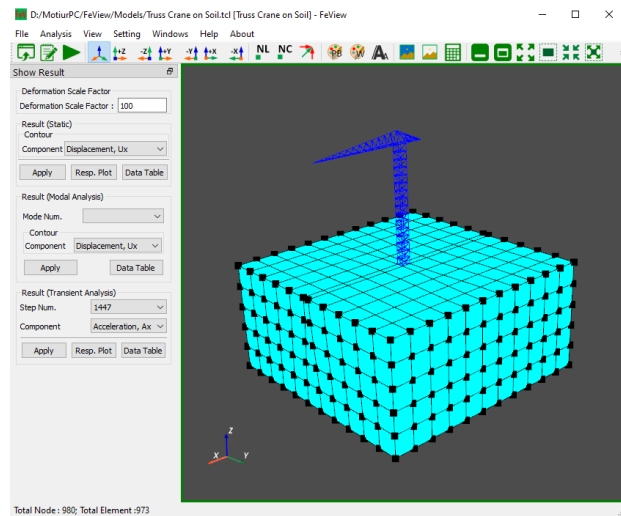


Figure: Truss crane on soil

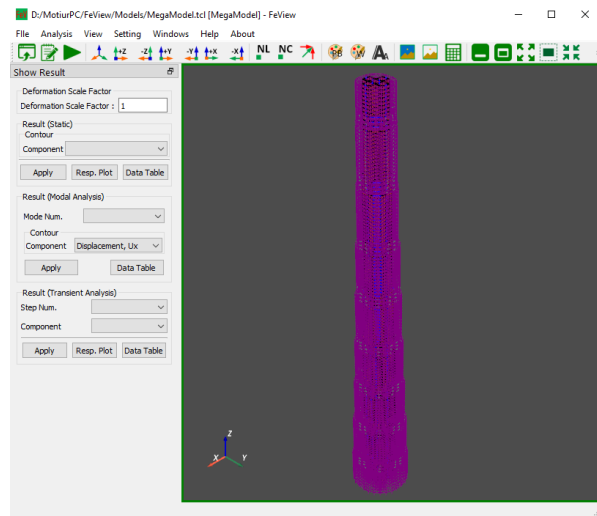


Figure: Mega-Tall Building Benchmark Models

2. Installation

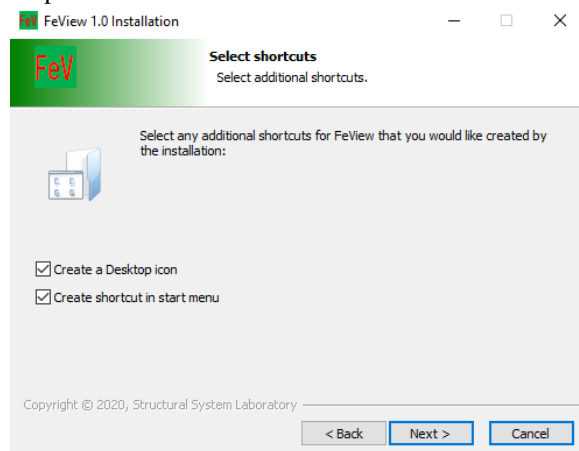
+ Follow the following steps:

Step-1: click "FeView_Setup.exe"

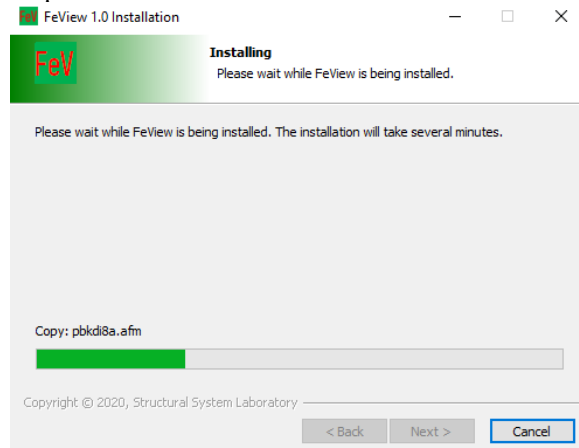
Step-2:



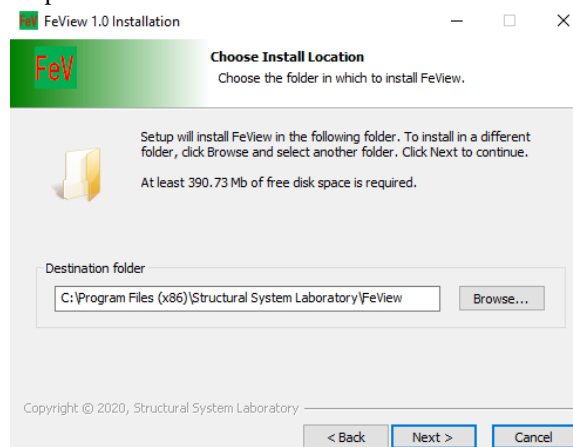
Step-4:



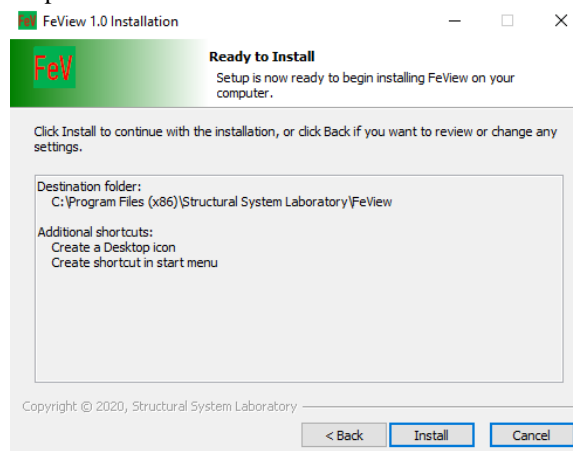
Step-6:



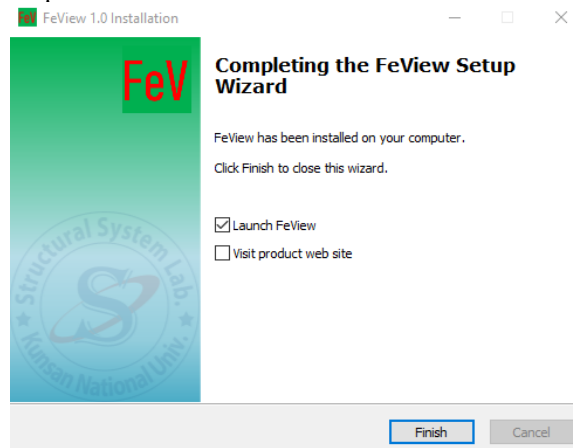
Step-3:



Step-5:



Step-7:



3. Active Tcl (required)

+ For running OpenSees in your system, you should have installed tcl/tk. If you already installed tcl/tk8.5 no need to installed again but if you don't have follow the following procedure

3.1 Download:

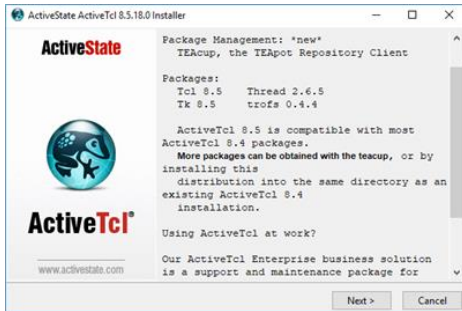
+ You can download tcl/tk from <https://www.activestate.com/products/tcl/downloads/>

3.2 Installation of tcl/tk

+ After downloading the Tcl/Tk executable, you will need to run it to install the DLL's on your computer.

+ Install by this way

Step 1:



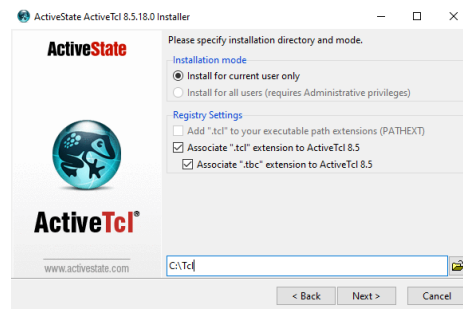
Step 2:



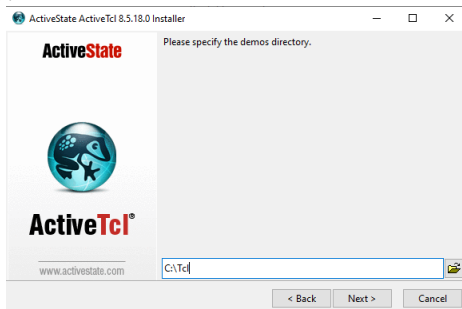
Step 3:



Step 4:



Step 5:



Step 6:



Step 7:



Step 8:



4. Features

+ Following features are included in this tools.

4.1 Analysis Type:

4.1.1 Static Analysis (linear/non linear)

- Gravity Analysis
- Pushover Analysis

4.1.2 Modal Analysis

4.1.3 Dynamic Analysis

4.2 Model Type:

- 2D and 3D structural frame/truss/shell/solid model

4.3 Materials:

- OpenSees supported all materials

4.4 Elements

4.4.1 Truss Elements:

- Truss Element
- Corotational Truss

4.4.2 Beam-Column Elements:

- Elastic Beam-Column
- Elastic Timoshenko Beam-Column
- Beam With Hinges Element
- Force-Based Beam-Column
- Displacement-Based Beam-Column
- Flexure-Shear Interaction Displacement-Based Beam-Column
- MVLEM - Multiple-Vertical-Line-Element-Model for RC Walls
- SFI_MVLEM - Cyclic Shear-Flexure Interaction Model for RC Walls

4.4.3 Link Elements:

- Two Node Link Element

4.4.4 Bearing Elements:

- Elastomeric Bearing (Plasticity) Element
- Elastomeric Bearing (Bouc-Wen) Element
- Flat Slider Bearing Element
- Single Friction Pendulum Bearing Element
- TFP Bearing
- Triple Friction Pendulum Element
- MultipleShearSpring Element
- KikuchiBearing Element
- YamamotoBiaxialHDR Element
- ElastomericX
- LeadRubberX
- HDR
- RJ-Watson EQS Bearing Element
- FPBearingPTV

4.4.5 Quadrilateral Elements:

- Quad Element
- Shell Element
- ShellDKGQ
- ShellNLDKGQ
- ShellNL
- Bbar Plane Strain Quadrilateral Element
- Enhanced Strain Quadrilateral Element
- SSPquad Element
- VS3D4
- AV3D4

4.4.6 Brick Elements:

- Standard Brick Element
- Bbar Brick Element
- SSPbrick Element
- Brick u-p Element
- eight-node 3D brick acoustic element
- eight-node zero-thickness 3D brick acoustic-structure interface element
- bbarBrickWithSensitivity

4.4.7 Cable Elements

- CatenaryCable Element

4.4.8 Tetrahedron Elements

- FourNodeTetrahedron

4.4.9 Triangular Elements

- Tri31 Element
- ShellDKGT
- ShellNLDKGT

5. Recorder Setting :

5.1 Node recorder

+ Example for node recorder setting:

- `recorder Node -file Node_displacements.out -time -nodeRange 1 220 -dof 1 2 3 disp`
- `recorder Node -file Node_rotations.out -time -nodeRange 1 220 -dof 4 5 6 disp`
- `recorder Node -file Node_forceReactions.out -time -nodeRange 1 220 -dof 1 2 3 reaction`
- `recorder Node -file Node_momentReactions.out -time -nodeRange 1 220 -dof 4 5 6 reaction`
- `recorder Node -file Node_accelerations.out -time -nodeRange 1 220 -dof 1 2 3 accel`
- `recorder Node -file Node_velocities.out -time -nodeRange 1 220 -dof 1 2 3 vel`

+ Please take care about nodeRange and output file name (must use specified file name which is given above like 'Node_displacements.out', 'Node_rotations.out' etc.)

5.2 Element recorder

+ Will be update soon...

6. Modal Analysis Setting

+ Please consider the following settings

6.1 Mode Number

+ Please set the mode number of modal analysis by the following way

- `set numModes 5`

6.2 Record eigenvectors

+ Please set the mode number of modal analysis by the following way

- `for { set k 1 } { $k <= $numModes } { incr k } { recorder Node -file [format "Mode_%i.out" $k] -nodeRange 1 525 -dof 1 2 3 "eigen $k" }`

7. Verification (Linear Static Analysis):

7.1 Two Dimensional Problem:

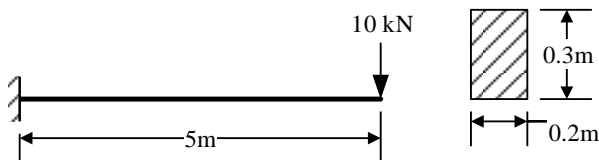
7.1.1 Plane Frame:

7.1.1.1 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:



Material Properties:

$E = 30 \text{ GPa}$
 $\nu = 0.2$

+ Results and Comparison

The most significant results are compared in the table below:

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

+ Comparison FeView & SAP2000 deform shape:

Shape Type	FeView	SAP2000
Deform shape		

+ Download TCL file:

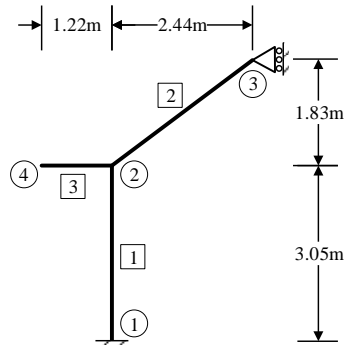
https://www.dropbox.com/s/6of2h6nvqus9mfq/Example_1.tcl?dl=0

7.1.1.2 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:



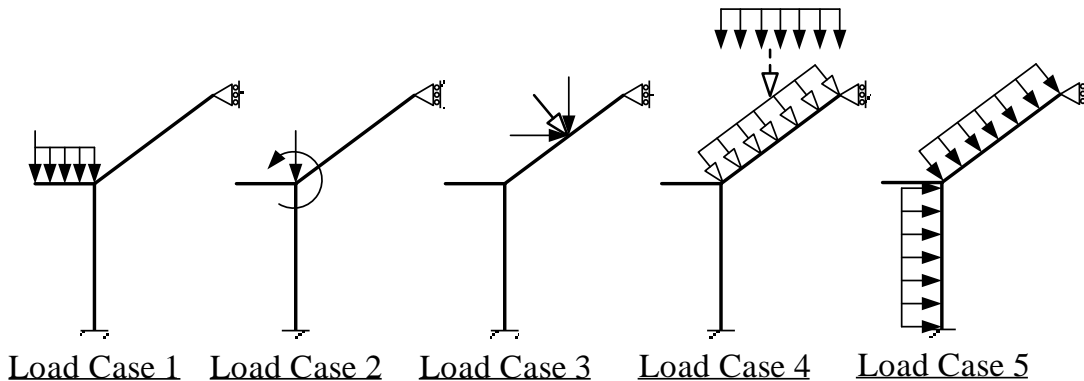
Material Properties:

$E = 24.81989 \text{ GPa}$

Section Properties:

$A = 0.0929 \text{ m}^2$

$I = 7.1925 \times 10^{-4} \text{ m}^4$



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 ($F_y = -26.27 \text{ kN/m}$) as the concentrated load is applied as point force in the global Y direction ($F_y = -44.48 \text{ kN}$))

• **Load Case 2** Global point force and point moment at joint 2 (force and moment are applied as permanent loads, in terms of forces ($F_y = -76.51 \text{ kN}$) and moments ($M_z = 73.75 \text{ kNm}$) 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, $F_x = 40.032 \text{ kN}$ and $F_z = -53.376 \text{ kN}$)

• **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, $F_x = -23.349 \cdot \cos\phi = -14.009 \text{ kN/m}$ and $F_y = -23.349 \cdot \sin\phi = -18.679 \text{ kN/m}$, respectively. Element angle $\phi = \arctan(1.83/2.44) = 36.9^\circ$

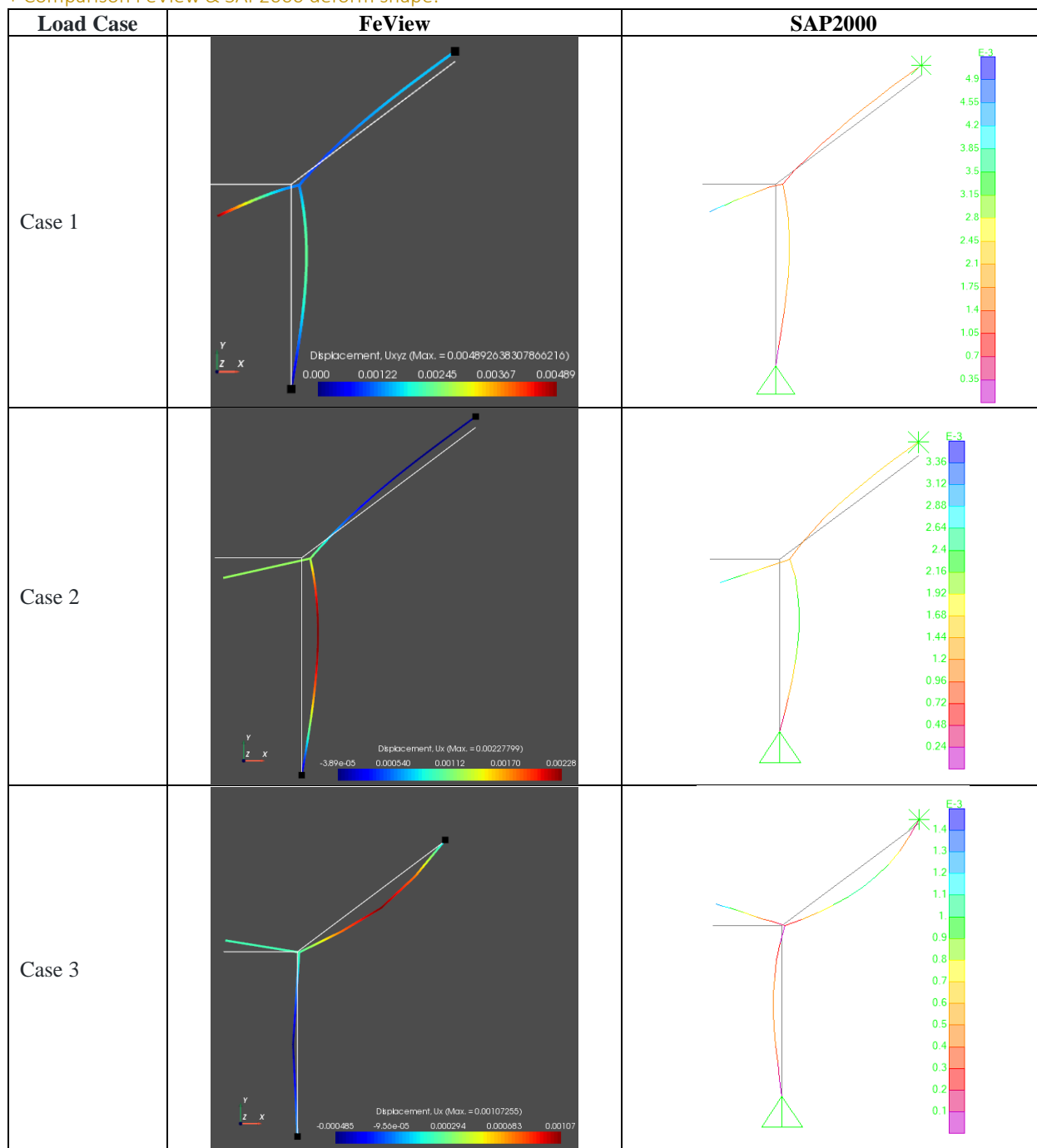
• **Load Case 5** : Distributed horizontal load 29.186 kN/m on frame element 1 (global X axis), applied on the element local y axis. Distributed horizontal load -29.186 kN/m applied on frame element 2 local y axis.

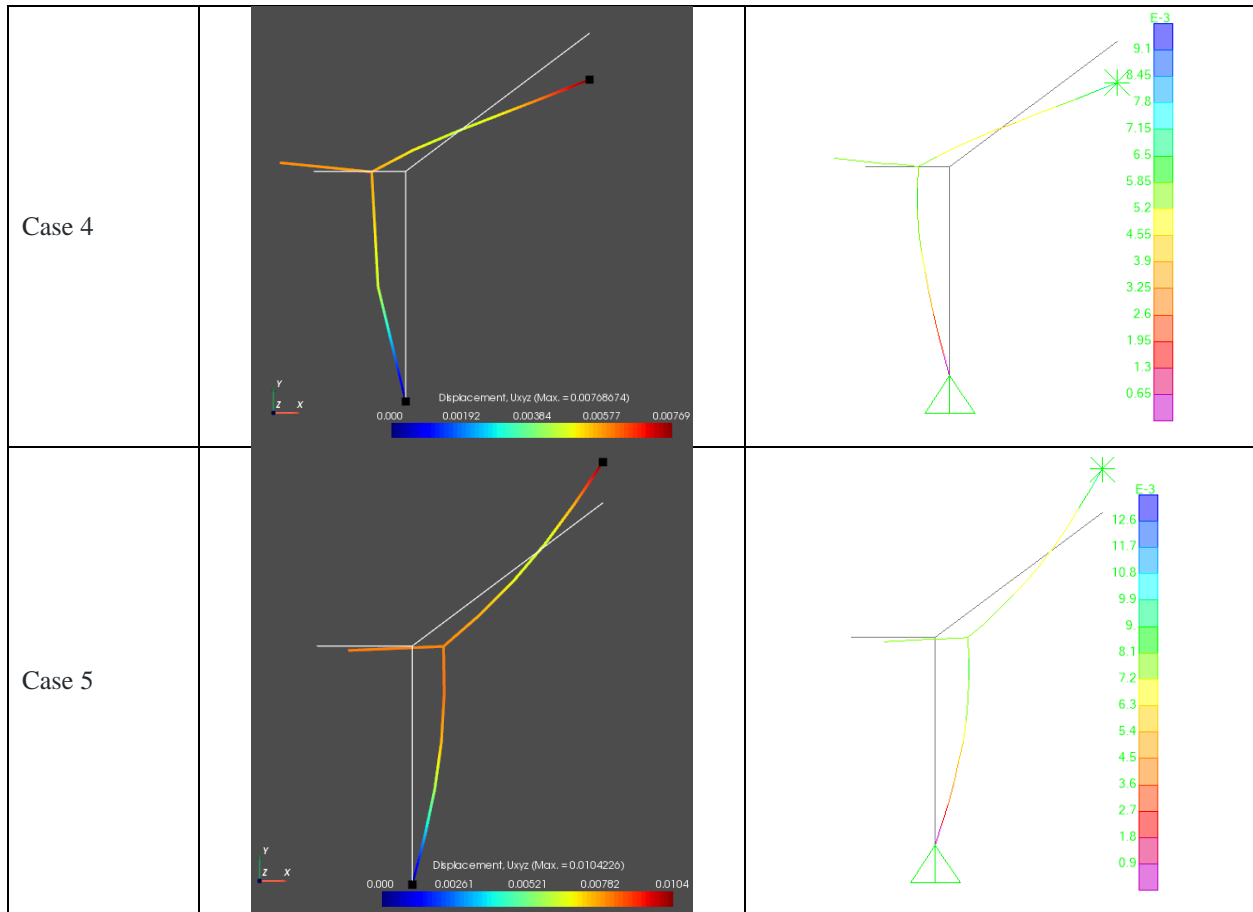
+ Results and Comparison

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

+ Comparison FeView & SAP2000 deform shape:





+ 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

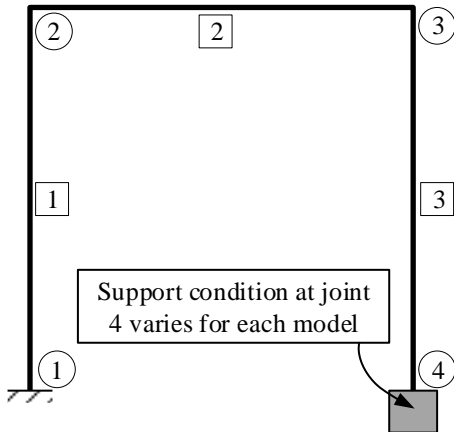
Load Case 5: https://www.dropbox.com/s/ckfyj843mzv2q0s/Example_2_LoadCase5.tcl?dl=0

7.1.1.3 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:



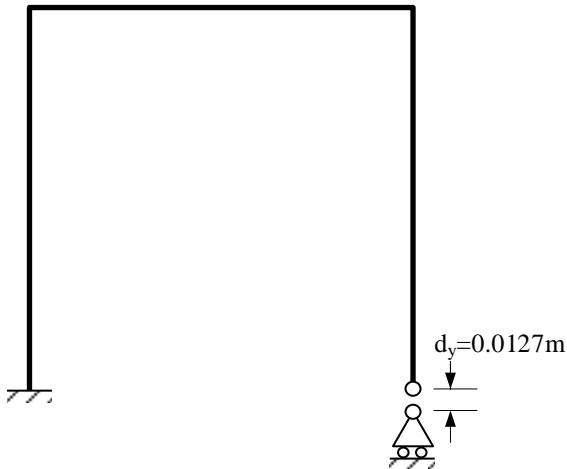
Material Properties:

$$E = 199.938 \text{ GPa}$$

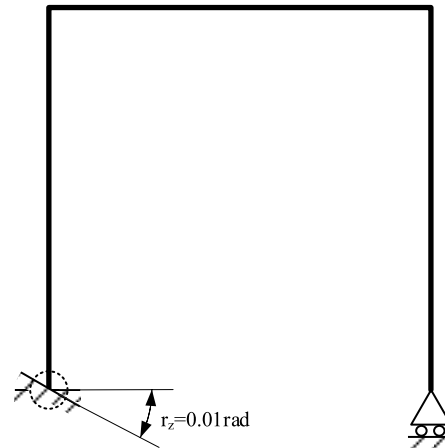
Section Properties:

$$A = 0.0929 \text{ m}^2$$

$$I = 7.19248 \times 10^{-4} \text{ m}^4$$



Model A



Model B

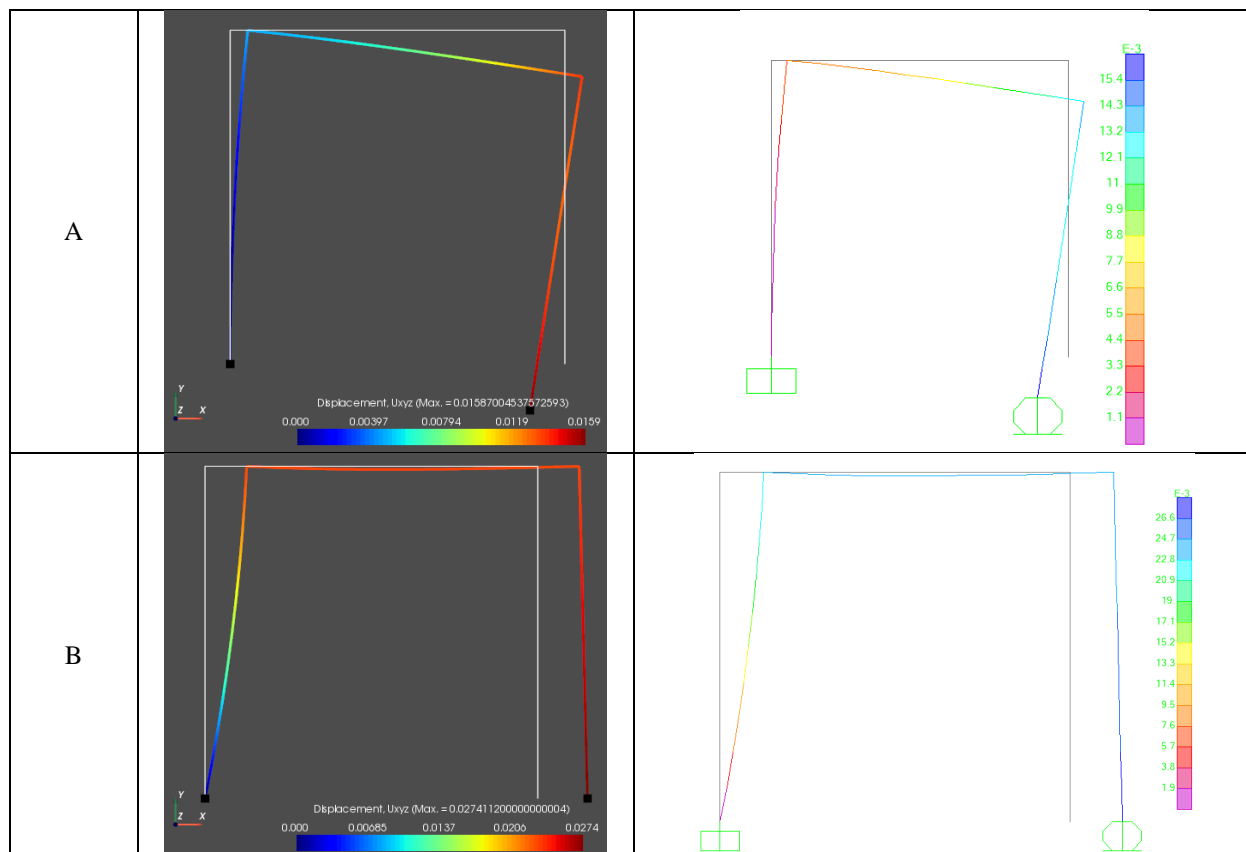
Results and Comparison

The most significant results are compared in the table below:

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

+ Comparison FeView & SAP2000 deform shape:

Model	FeView	SAP2000
-------	--------	---------



+ Download TCL file:

Model A: https://www.dropbox.com/s/6iiv7r22rqswmf/Example_3A.tcl?dl=0

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

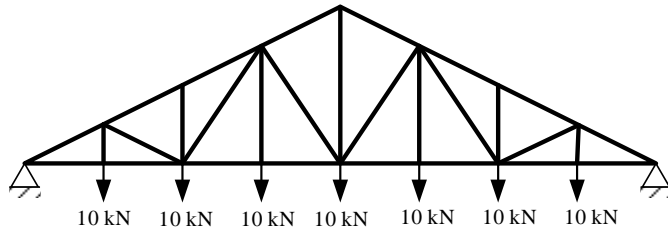
7.1.2 Plane Truss:

7.1.2.1 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:



Material Properties:

$$E = 200 \text{ GPa}$$

$$\nu = 0.2$$

Section Properties:

$$A = 0.002 \text{ m}^2$$

+ Results and Comparison

The most significant results are compared in the table below:

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

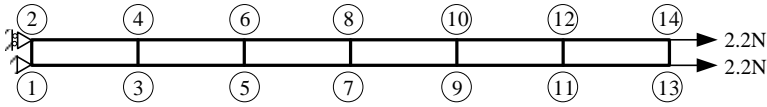
7.1.3 Planes:

7.1.3.1 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:

$E = 68.95 \text{ GPa}$

$\nu = 0.3$

$G = 26.52 \text{ GPa}$

Section Properties:

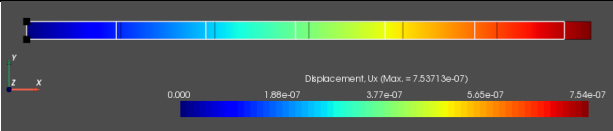
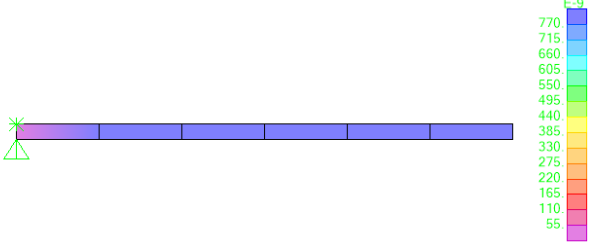
Plane element thickness = $2.54 \times 10^{-3} \text{ m}$

+ Results and Comparison

The most significant results are compared in the table below:

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

7.1.3.2 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 U_x and U_y 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

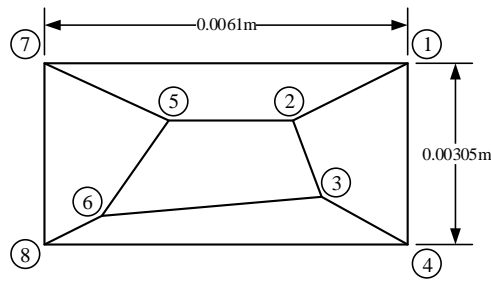
Material Properties:

$$E = 68.95 \text{ GPa}$$

$$\nu = 0.25$$

Section Properties:

$$\text{Plane element thickness} = 2.54 \times 10^{-3} \text{ m}$$



+ Loading

The loading is provided in the form of prescribed edge displacements U_x and U_y , which are imposed on joints 1, 4, 7 and 8. Those displacements are defined by the following equations.

$$U_x = \frac{x + \frac{y}{2}}{1000}, \quad U_y = \frac{y + \frac{x}{2}}{1000}$$

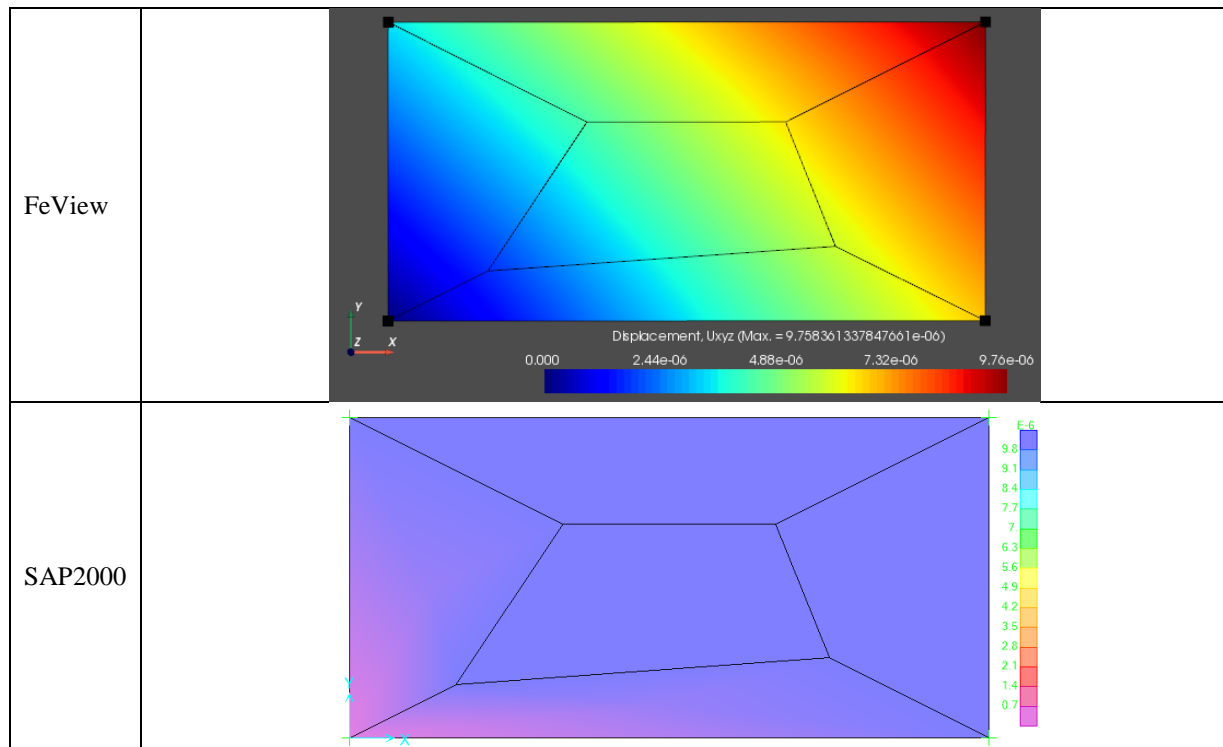
+ Results and Comparison

The most significant results are compared in the table below:

Joint	FeView		SAP2000	
	U_x	U_y	U_x	U_y
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
----------	--------------



+ Download TCL file:

https://www.dropbox.com/s/xp6ioyypotu9c1h/Example_6.tcl?dl=0

7.2 Three Dimensional Problem:

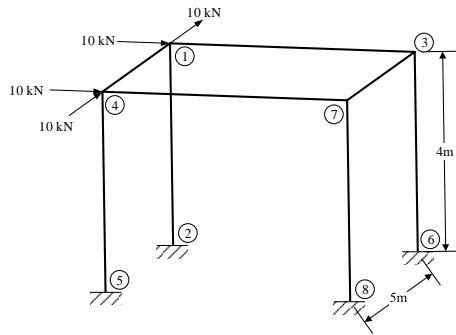
7.2.1 Space Frame:

7.2.1.1 Verification Example 7:

A space frame is subjected to load on X and Y direction as shown in figure. The displacements (U_x , U_y , U_z) 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:

$$E = 30 \text{ GPa}$$

$$\nu = 0.2$$

Section Properties:

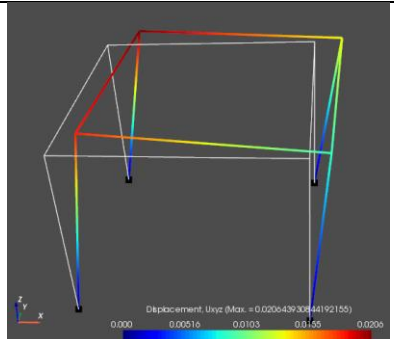
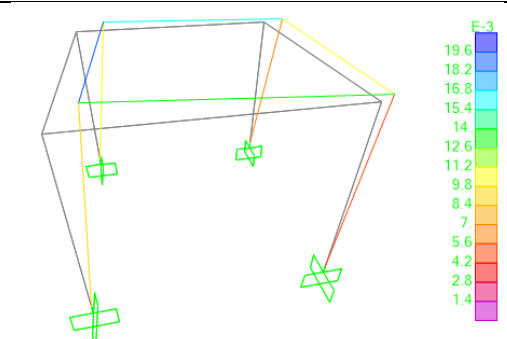
$$I = 1.33 \times 10^4 \text{ m}$$

+ Results and Comparison

The most significant results are compared in the table below:

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

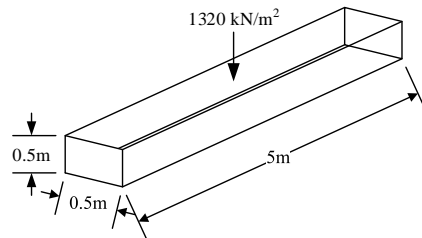
7.2.2 Solid:

7.2.2.1 Verification Example 8:

A Solid simple beam is subjected to load on X and Y direction as shown in figure. The displacements (U_x , U_y , U_z) 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:

$$E = 30 \text{ GPa}$$

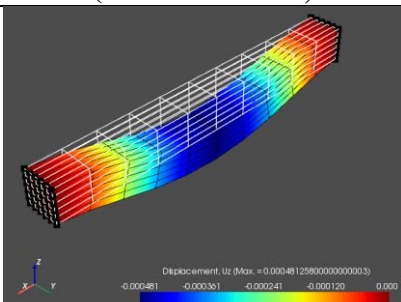
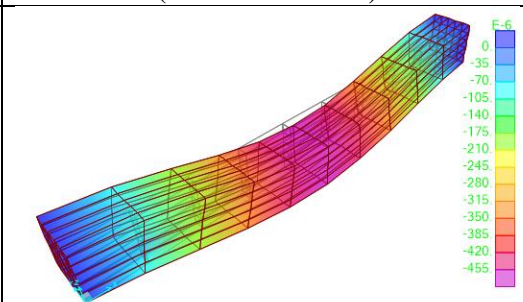
$$\nu = 0.2$$

+ Results and Comparison

The most significant results are compared in the table below:

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

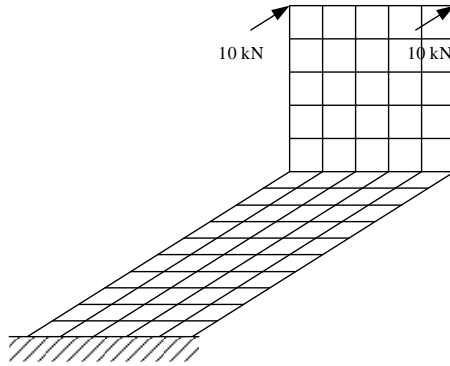
7.2.3 Shell:

7.2.3.1 Example 9:

A steel angle modelled by shell element is loaded as shown in figure. The maximum displacements in 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:

$E = 200 \text{ GPa}$

$\nu = 0.3$

Section Properties:

Shell thickness = 0.02 m

+ Results and Comparison

The most significant results are compared in the table below:

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

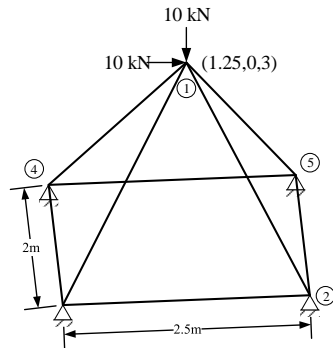
7.2.4 Space Truss:

1.2.4.1 Verification Example 10:

A steel space truss is loaded in X and Z direction at joint 1.. The displacements in 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:

$E = 200 \text{ GPa}$

Section Properties:

$A = 0.001 \text{ m}^2$

+ Results and Comparison

The most significant results are compared in the table below:

Output Parameter	FeView	SAP2000
$U_{x,(\text{node } 1)} \text{ [m]}$	3.40E-4	3.40E-4
$U_{z,(\text{node } 1)} \text{ [m]}$	-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/jla5tp5yjp8xm8j6/Example_10.tcl?dl=0

8. Verification (Modal Analysis):

8.1 Two Dimensional Problem:

8.1.1 Plane Frame:

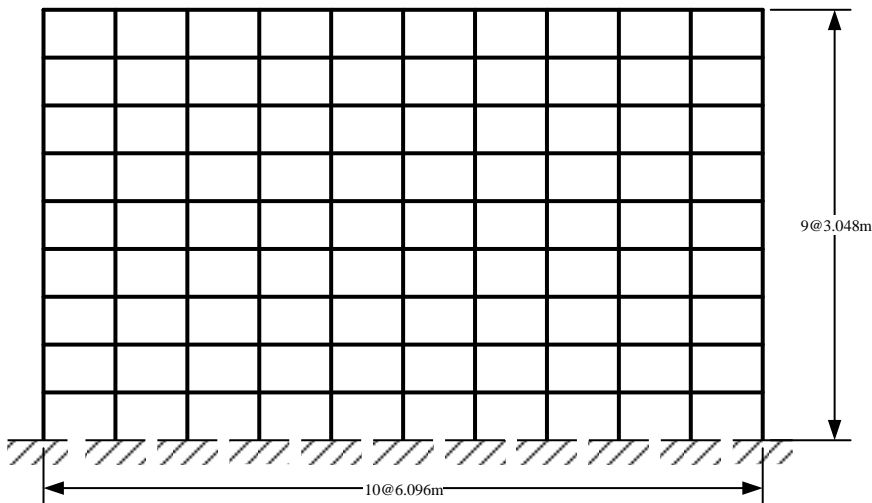
8.1.1.1 Verification Example 11:

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:

$E = 20.6832410 \text{ GPa}$

Section Properties:

$A = 0.279 \text{ m}^2$

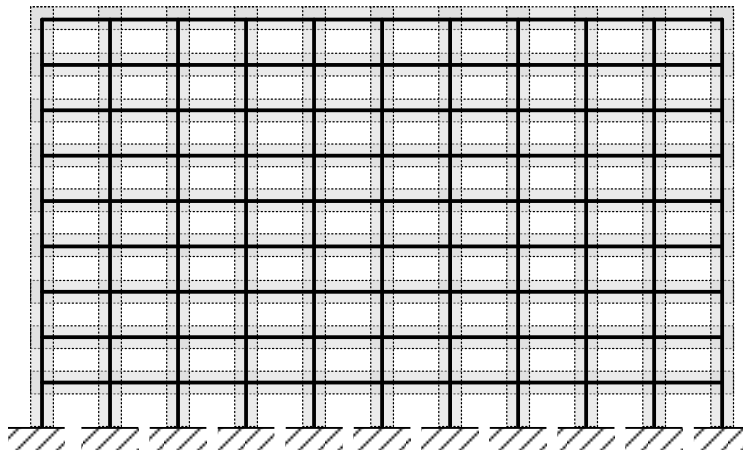
$I = 0.008631 \text{ m}^4$

+ Loading/Modeling

The frame objects are modeled through elastic frame elements with specified Mass/Length³ = 514.81 ton/m³.

All the base nodes are fully restrained.

The FE model is presented below:

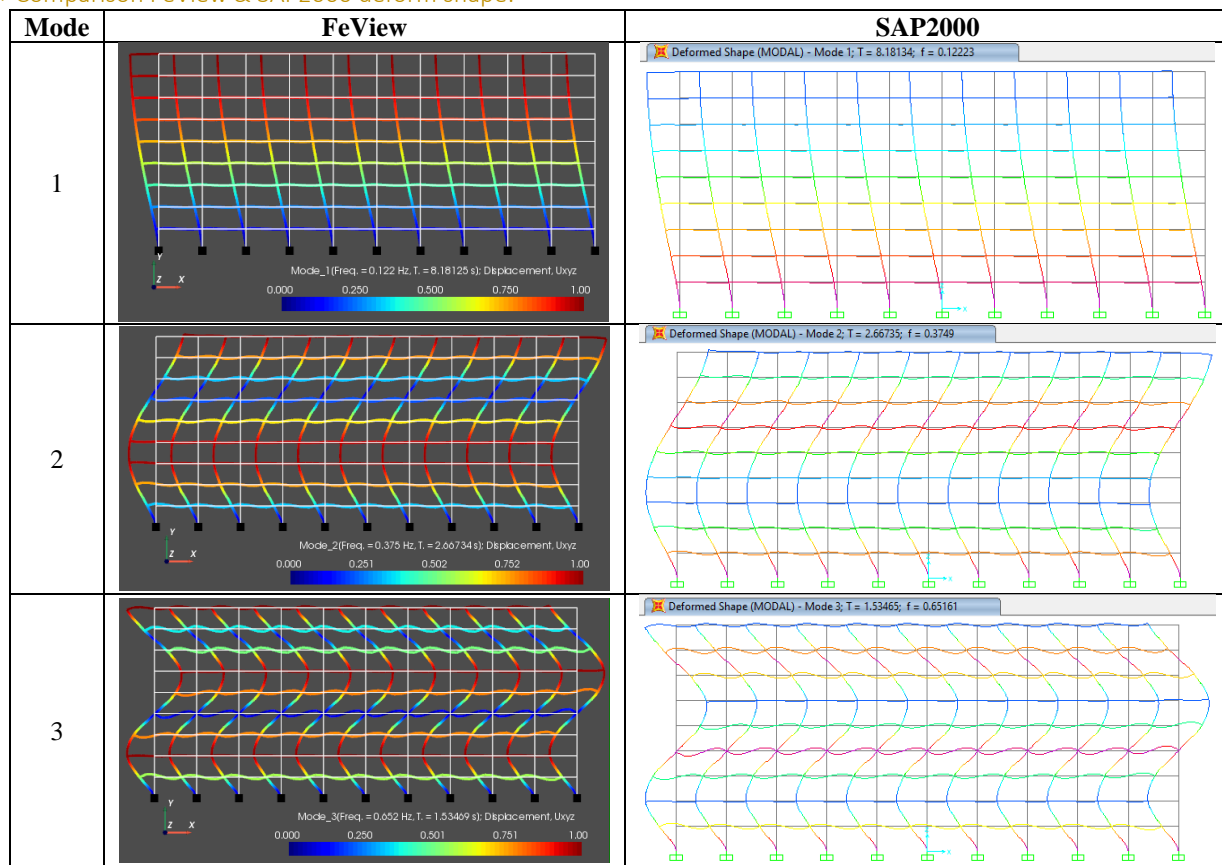


+ Results and Comparison

The most significant results are compared in the table below:

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

+ Comparison FeView & SAP2000 deform shape:



+ Download TCL file:

https://www.dropbox.com/s/v09o6t13w8e7uw2/Example_11.tcl?dl=0

8.1.2 Plane Truss:

8.1.2.1 Verification Example 12:

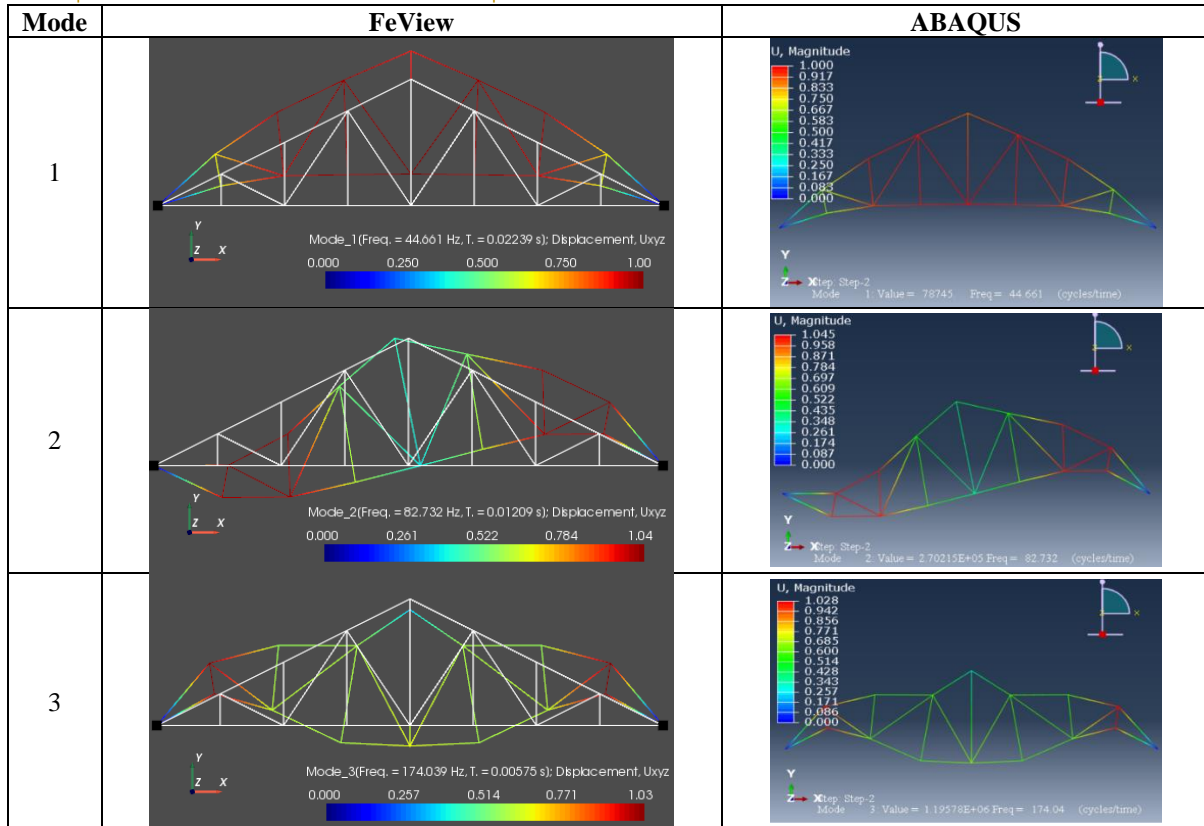
Example 4 with material density 7850 kg/m³ is considered for this example analysed for the first three mode. The results obtained with the FE analysis program FeView are compared with ABAQUS results.

+ Results and Comparison

The most significant results are compared in the table below:

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

+ Comparison FeView & SAP2000 deform shape:



+ Download TCL file:

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

8.1.3 Plane:

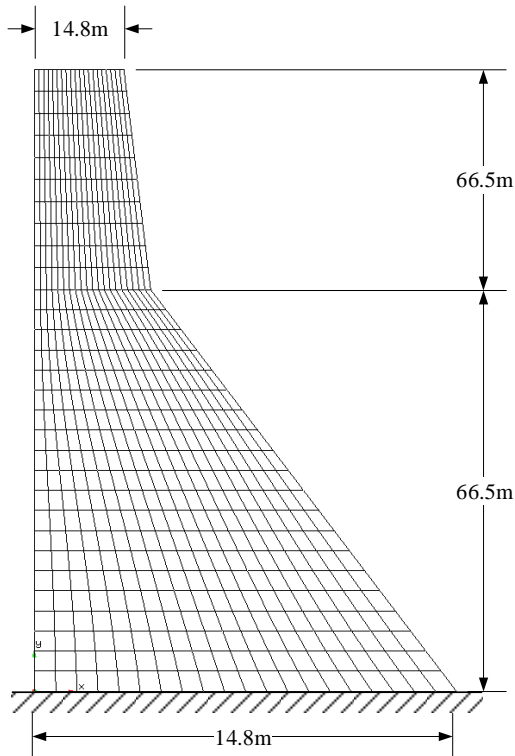
8.1.3.1 Verification Example 13:

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:



Material Properties:

$$E = 31.027 \text{ GPa}$$

$$\nu = 0.15$$

$$\rho = 2643 \text{ kg/m}^3$$

Section Properties:

Plane element thickness = 1 m

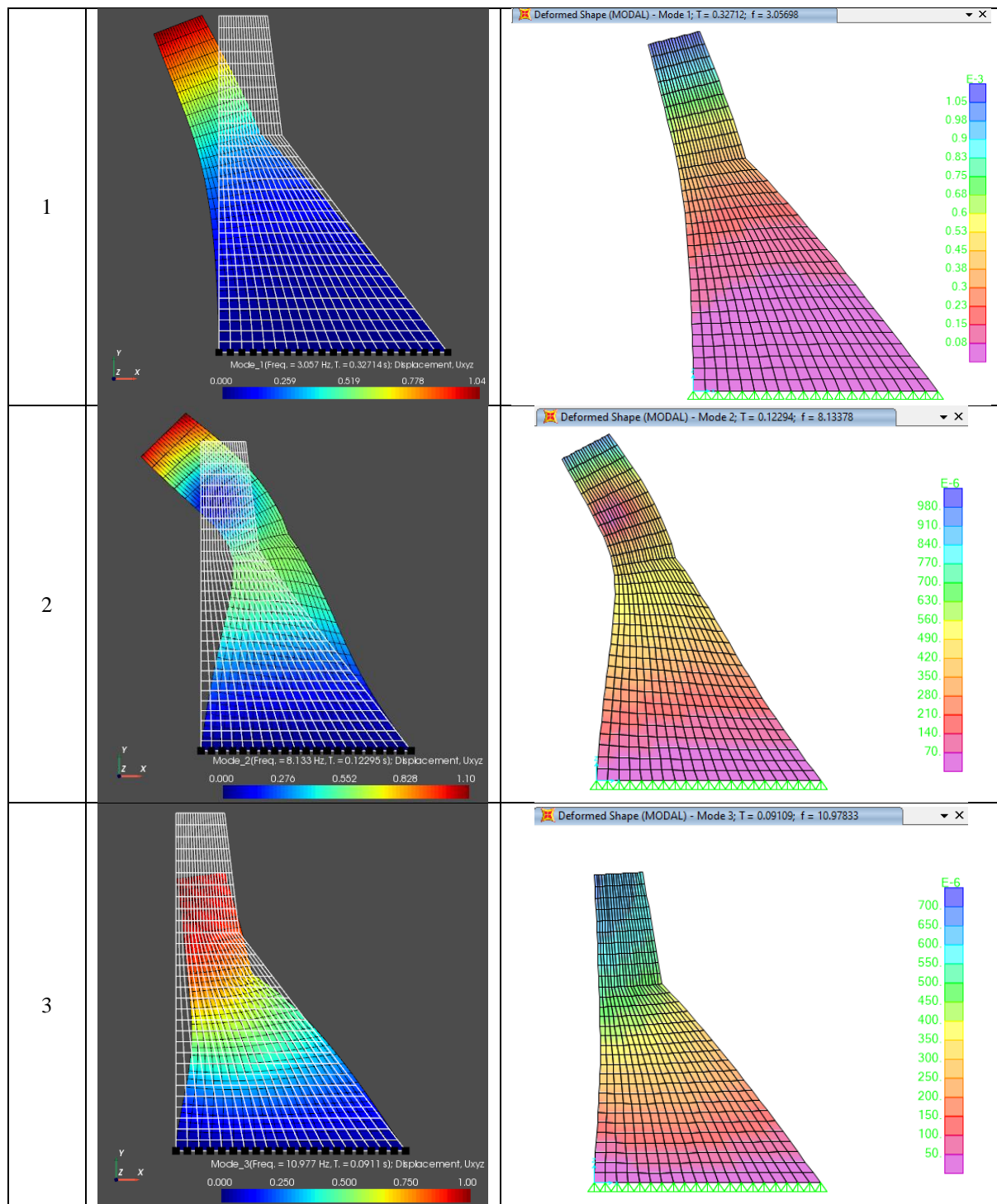
+ Results and Comparison

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
------	--------	---------



+ Download TCL file:

https://www.dropbox.com/s/s7jnuychejwfs0/Example_13.tcl?dl=0

8.2 Three Dimensional Problem:

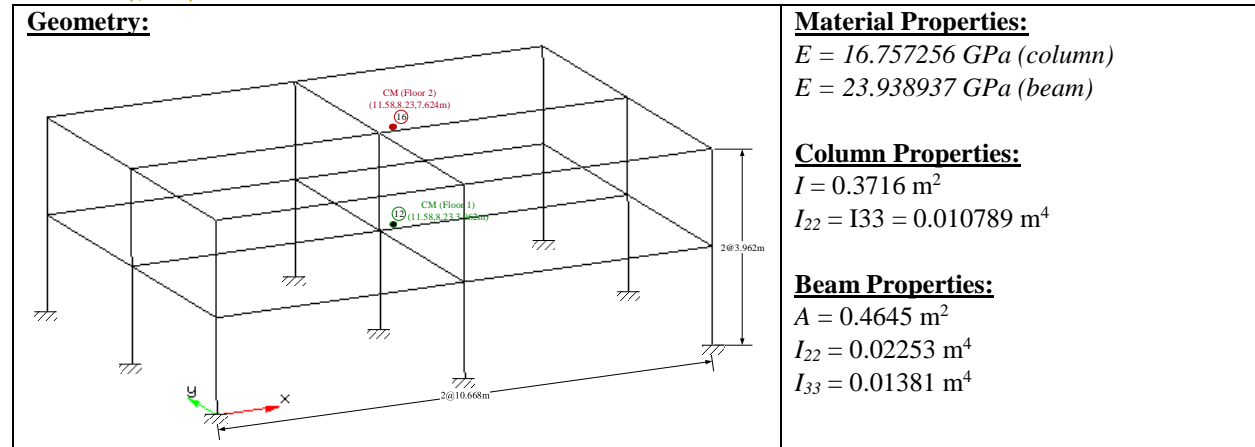
8.2.1 Space Frame:

8.2.1.1 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.

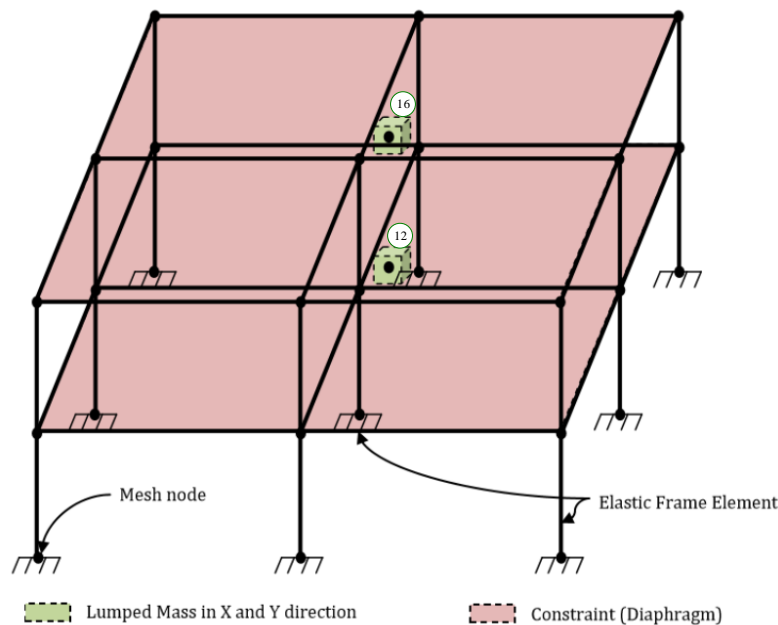
+ Geometry, Properties



+ 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:

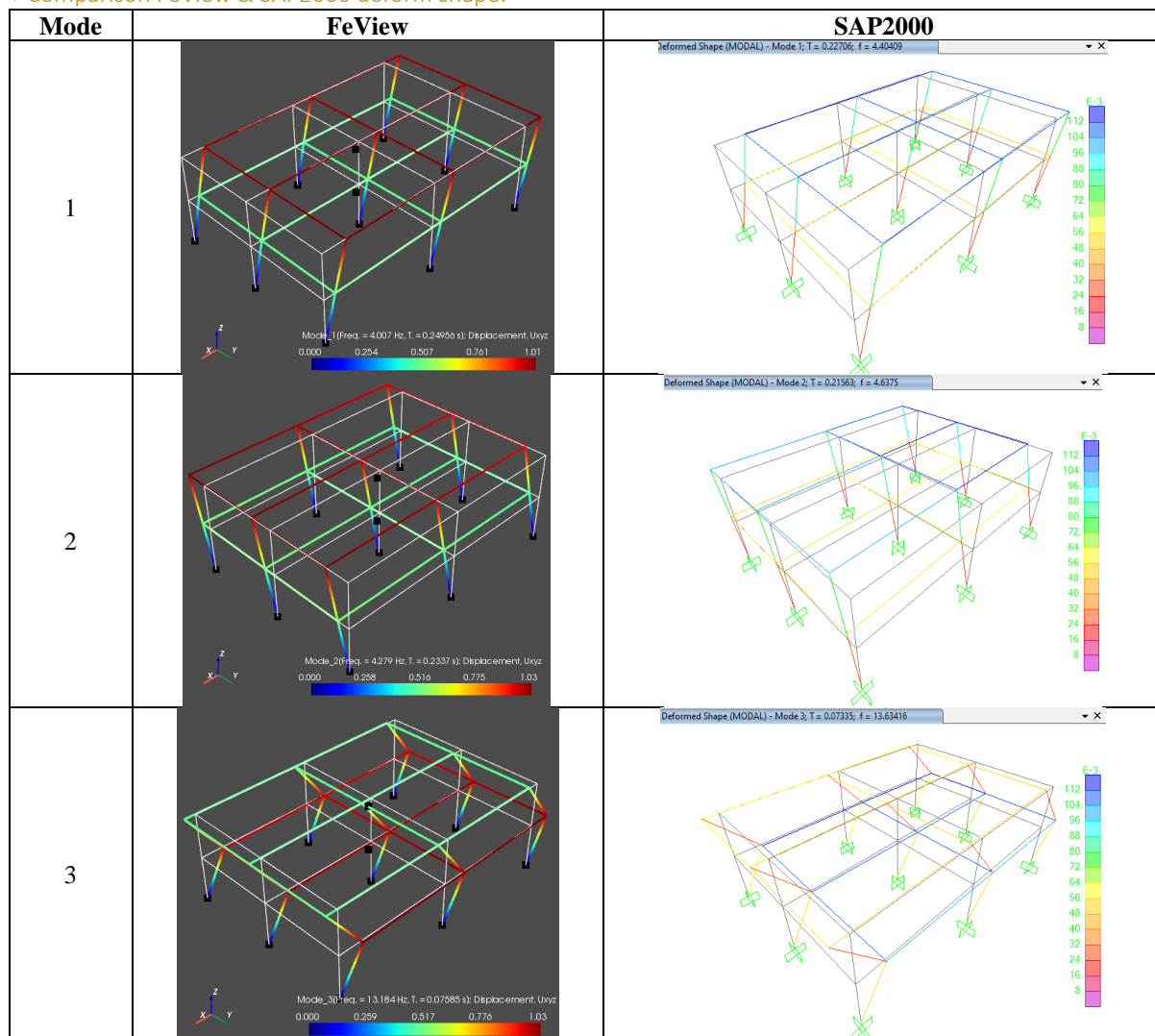


+ Results and Comparison

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:



+ Download TCL file:

https://www.dropbox.com/s/9utmgm698dhpud/Example_14.tcl?dl=0

8.2.2 Solid:

8.2.2.1 Verification Example 15:

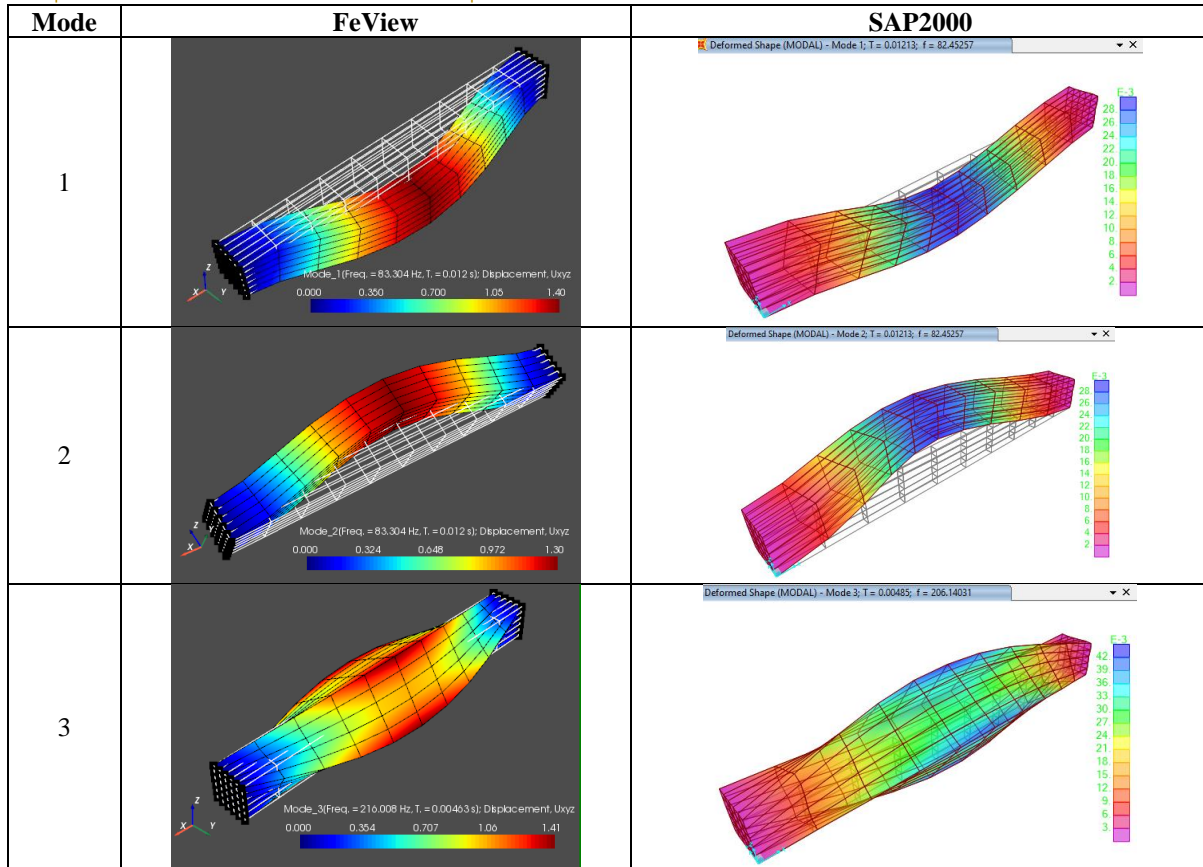
Example 8 with material density 2400 kg/m³ is considered for this example and analysed for the first three modes. The results obtained with the FE analysis program FeView are compared with SAP2000 results..

+ 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:



+ Download TCL file:

https://www.dropbox.com/s/41e1cxy8kzhavj/Example_15.tcl?dl=0

8.2.3 Shell:

8.2.3.1 Verification Example 16:

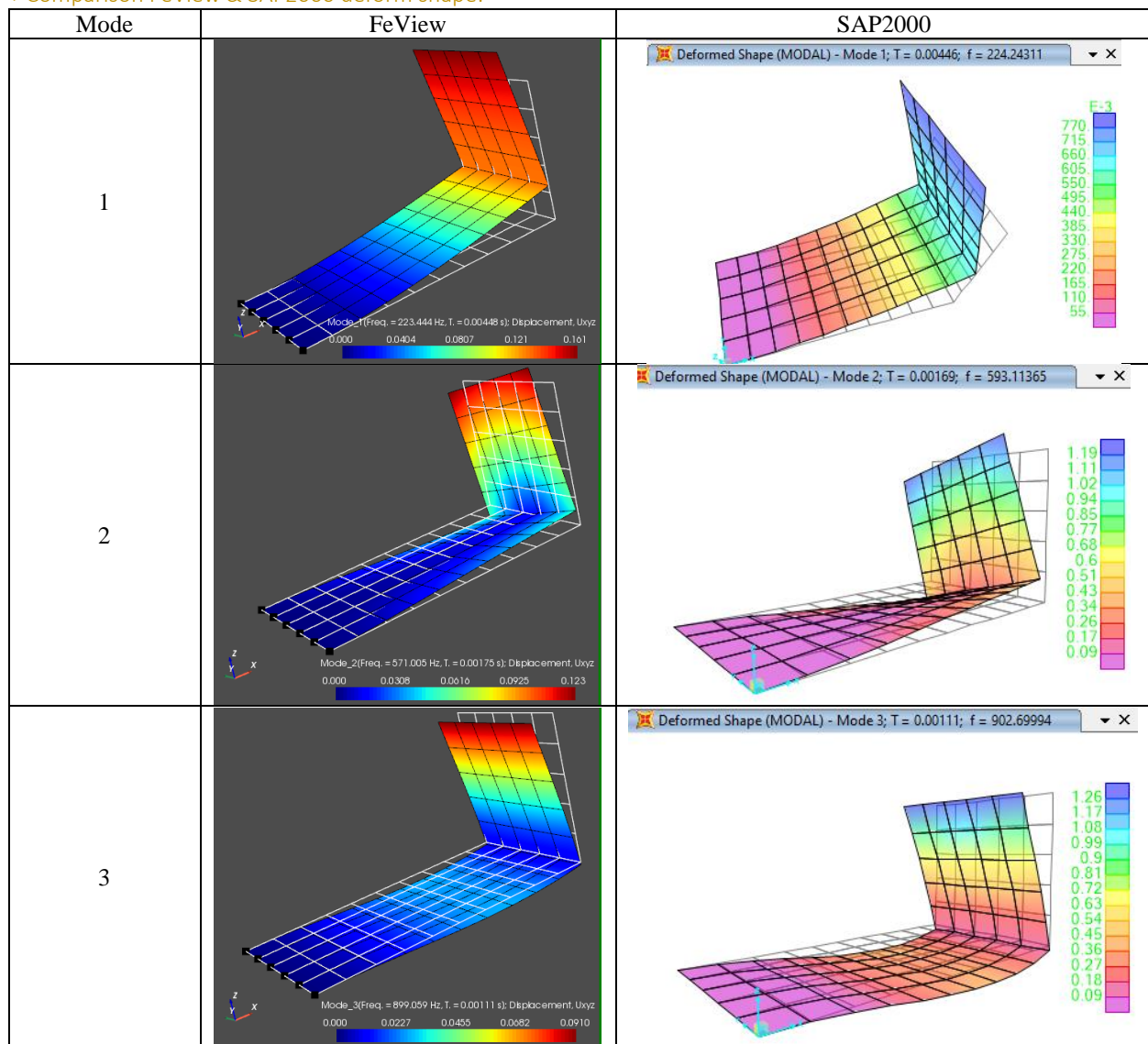
Example 9 with material density 7850 kg/m³ is considered for this example and analysed for the first three modes. The results obtained with the FE analysis program FeView are compared with SAP2000 results..

+ Results and Comparison

The most significant results are compared in the table below:

Output Parameter	FeView	SAP2000
Period T_1 [s] (1st mode)	0.0045	0.0045
Period T_2 [s] (2nd mode)	0.0018	0.0017
Period T_3 [s] (3rd mode)	0.0011	0.0011

+ Comparison FeView & SAP2000 deform shape:



+ Download TCL file:

https://www.dropbox.com/s/lcdhco2f4q6rn4l/Example_16.tcl?dl=0

8.2.4 Space Truss:

8.2.4.1 Verification Example 17:

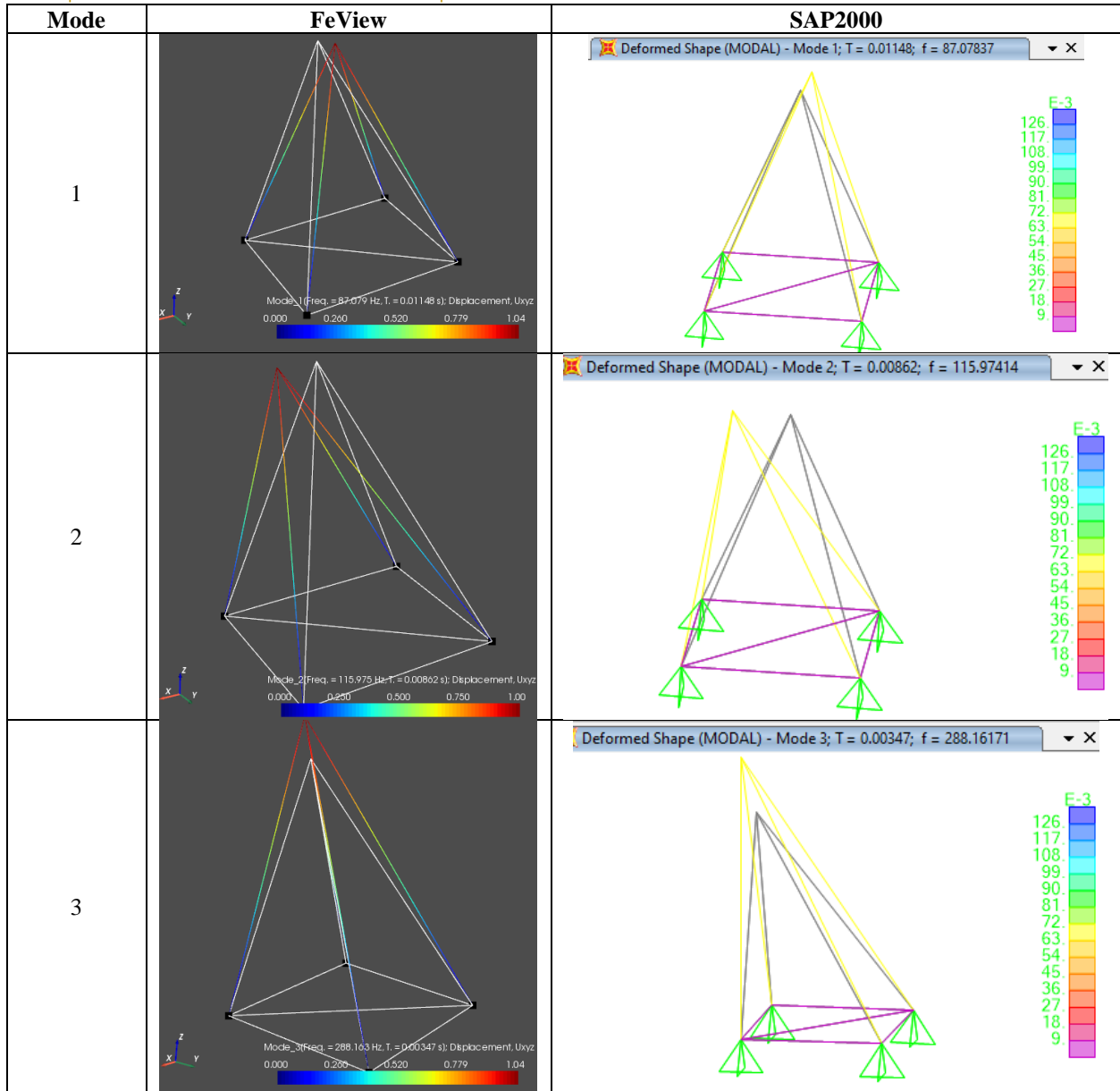
Example 10 with material density 7850 kg/m³ is considered for this example and analysed for the first three modes. The results obtained with the FE analysis program FeView are compared with SAP2000 results..

+ 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:



+ Download TCL file:

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

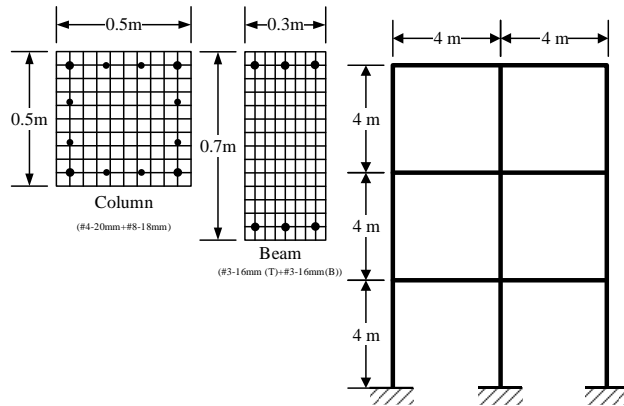
9. Example:

9.1 Static Pushover Analysis (Non-linear Analysis):

+ Static pushover analysis (non-linear) of inelastic plane RCC shown in figure:

+ **Geometry, Properties**

Geometry:



Materials properties:

Confined Concrete:

$$f_{pc} = -20 \text{ MPa}$$

$$\epsilon_{psc0} = -0.002$$

$$f_{pcu} = -17 \text{ MPa}$$

$$\epsilon_{psc u} = -0.0035$$

Unconfined Concrete:

$$f_{pc} = -20 \text{ MPa}$$

$$\epsilon_{psc0} = -0.002$$

$$f_{pcu} = -17 \text{ MPa}$$

$$\epsilon_{psc u} = -0.005$$

Steel:

$$f_y = 500 \text{ MPa}$$

$$E_s = 200 \text{ GPa}$$

$$b = 0.15$$

$$a_1 = 0$$

$$a_2 = 1$$

$$a_3 = 0$$

$$a_4 = 1$$

+ **Loading/Modeling (for making TCL file)**

- Materials:

→ Confined Concrete: concrete01

→ Unconfined Concrete: concrete01

→ Rebar: steel01


- Element:

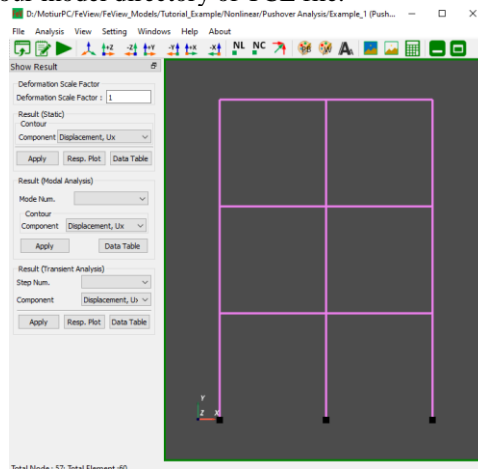
→ Beam/Column: forceBeamColumn with fiber element

- Loading:


→ Static load: -15 kN/m element uniform gravity forces to all beam

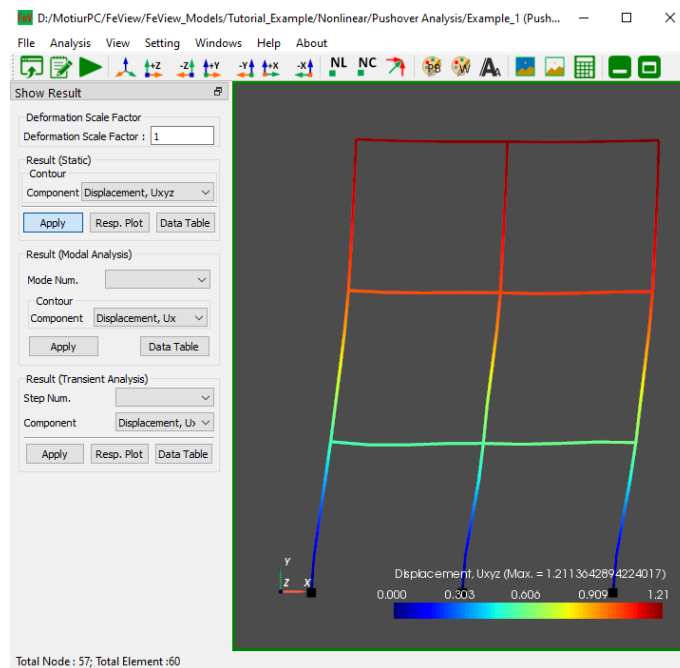
+ **Open Model:**

After clicking  button browse your model directory of TCL file.

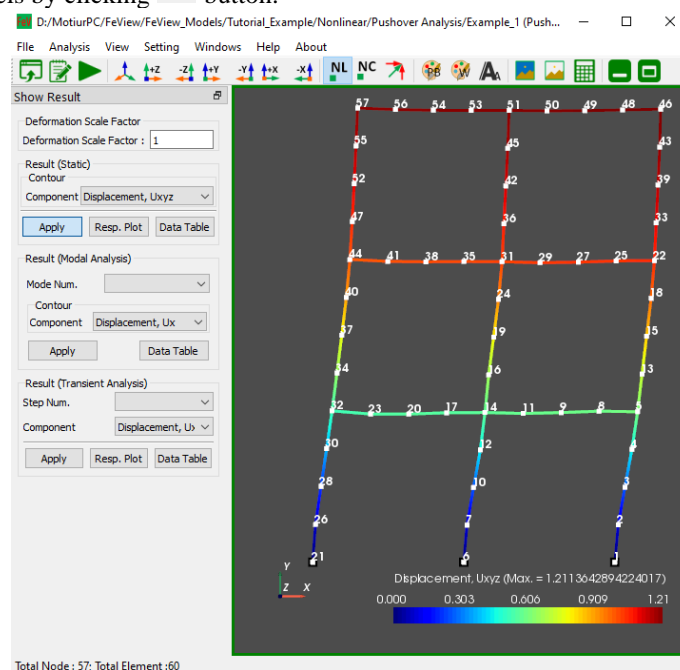


+ **Run OpenSees and show deformed model:**

Click  to run the model and then click "Apply" button.

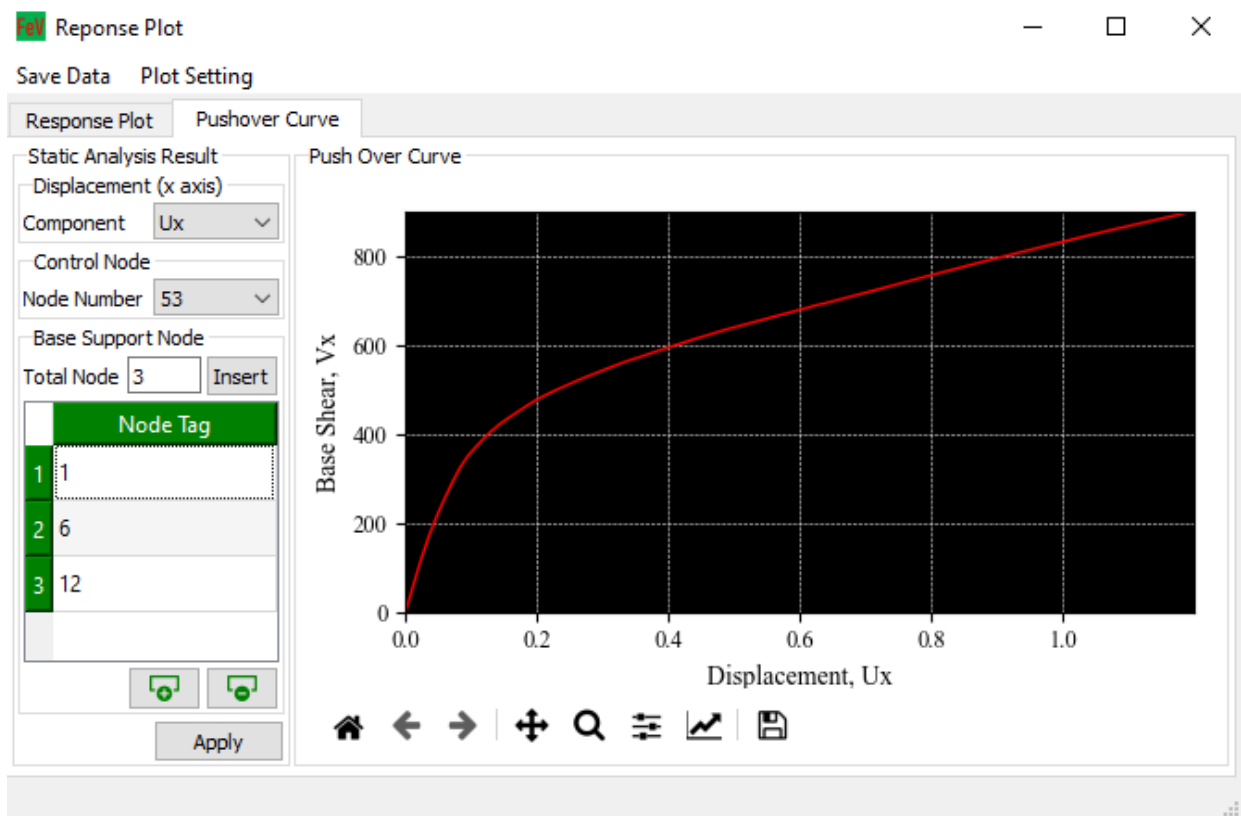


You can see the node labels by clicking **NL** button.



+ Pushover Curve:

- Step1: Click “Resp. Plot” button and the go “Pushover Curve” tab
- Step 2: Set displacement component and control node
- Step 3: Set base support nodes
- Step 4: click “Apply button”



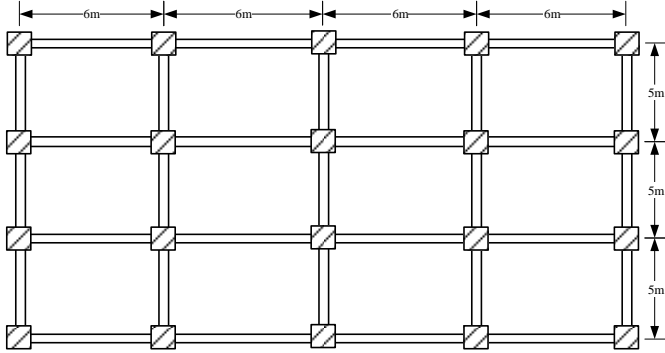
+ Download TCL file:

Download link:


9.2 Dynamic Analysis:

+ Modal and transient analysis of high rise building (elastic moment frame) shown in figure:

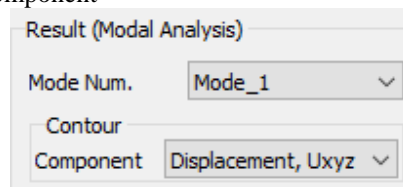
+ Geometry, Properties

<p>Geometry:</p> 	<p>Material Properties:</p> <p>$E = 30 \text{ GPa}$</p> <p>Column Size:</p> <p>$b \times h = 0.6 \text{ m} \times 0.6 \text{ m}$</p> <p>Beam Size:</p> <p>$b \times h = 0.3 \text{ m} \times 0.5 \text{ m}$</p> <p>Story:</p> <p>Number: 15</p> <p>Story Height: 3m</p>
---	--

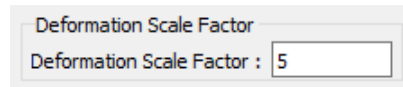
+ Mode Shape:

Step 1: After clicking  button browse your model directory of TCL file.

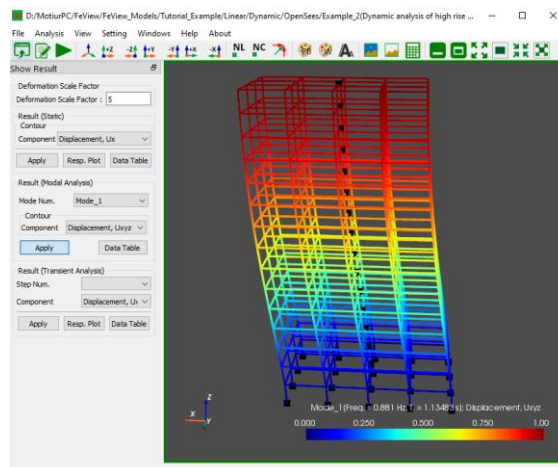
Step 2: Set Mode Num. and contour component



Step 4: if needed you can select



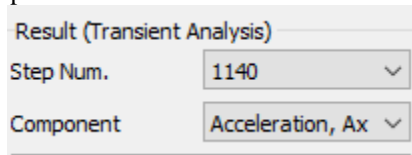
Step 3: Click “Apply” Button



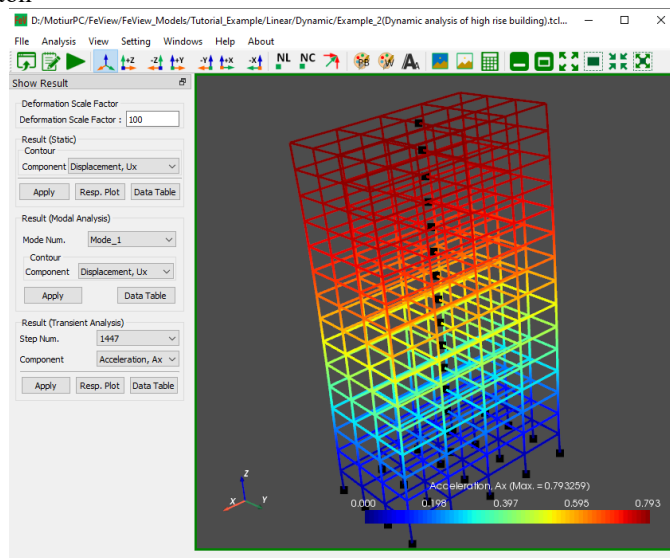
+ Deformed shape for transient analysis:

Step 1: Set step number (in which step you want to see the deform shape). If step number is disappear in the combo box click “Apply” button in static analysis.

Step 2: Set the component for contour plot



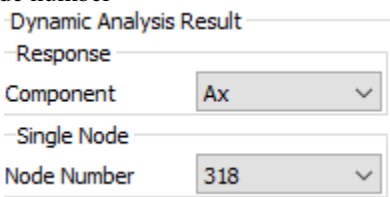
Step 2: Click “Apply” button



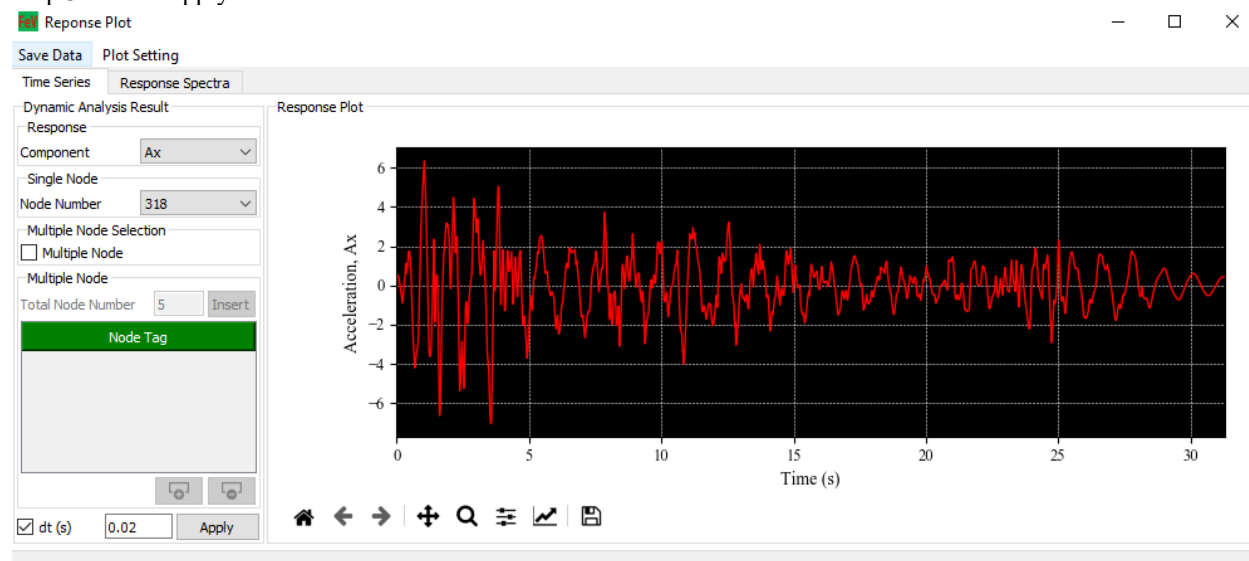
+ Response plot for transient analysis:

Step1: After clicking “Apply” button click “Resp. Plot” and then select “Time Series” tab

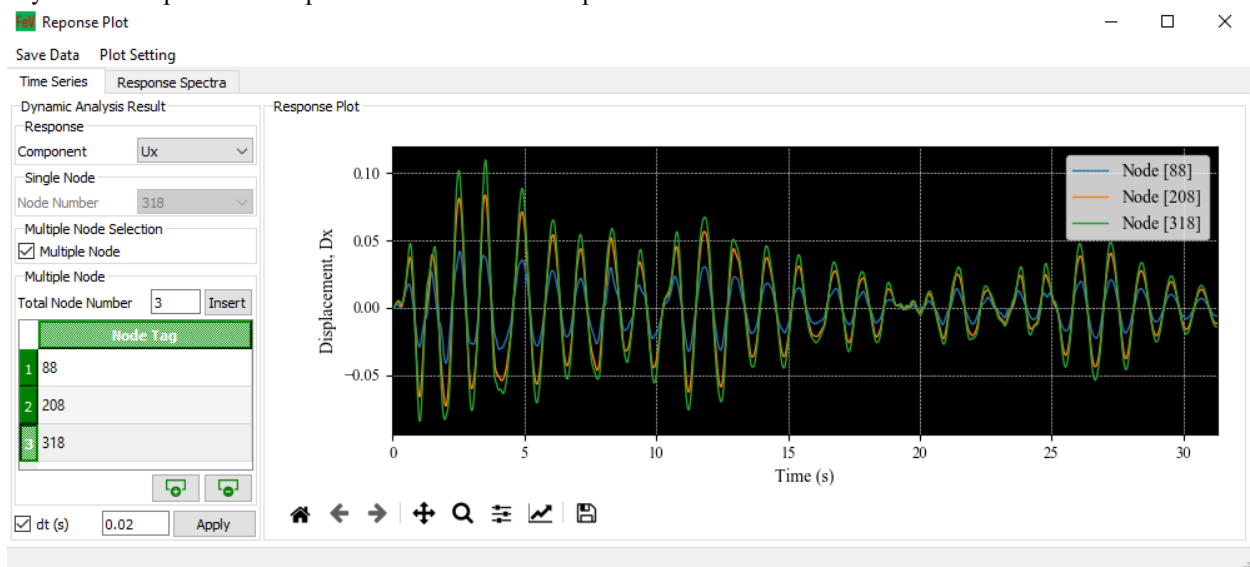
Step 2: Set response component and node number



Step 3: click “Apply button”



If you want to plot for multiple node check the “Multiple Node” box



+ Response Spectrum:

Set the following values and then click “Apply” button

Dynamic Analysis Result

Single Node

Node Number

Component

☒ X ☐ Y ☐ Z

Time Increment, dt

Damping Value

Horizontal Axis

☒ Period ☐ Frequency

Vertical Axis

☒ PSa ☐ PSv ☐ Sd

