Seismo-VLAB's DEBUGGING REPORT

Danilo S. Kusanovic, Elnaz E. Seylabi, and Domniki Asimaki.

October 25, 2021

Abstract

Seismo-VLAB (**SVL**) is a simple, fast, and extendable C++ multi-platform finite element software designed to run large-scale simulations of dynamic, nonlinear soil-structure interaction problems. In this report several debugging cases are presented in order to verify the accuracy and well-behaviour of the implemented features. The DEBUG CASE's names are as follows: **L01-Analysis_Formulation_Comment_Material_Element**, where **L** is a letter that denotes complexity, **Analysis** can be ST=Static, or DY=Dynamic, **Formulation** can be Lin=Linearized or Kin=kinematics, **Comment** is a description, **Material** and **Element** are the **SVL**'s class.

DEBUG CASE : A01-DY_Lin_2D_Elastic_ZeroLength

The problem showed in Figure 1 is defined to test `ZeroLength1D` element with material type `Elastic1DLinear`. The material has a elasticity moduli $E = 200\ GPa$, and a Poisson's ratio $\nu = 0.25$. Both nodes (1) and (2) have coordinate $(x, y) = (0.0, 0.0)$. Node (1) is fixed in X and Y directions, while node (2) is fixed in Y direction. For dynamic analysis, the nodal stress applied at node (2) is defined as $\sigma(t) = 107.5 \cdot t\ \sin(2\pi\, t)\ Pa$. Responses are verified against analytical solution. Figure 2 shows the strain, stress, and material constitutive responses at (2).

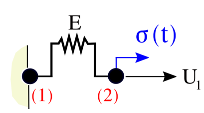


Figure 1: Varification for `ZeroLength1D` with `Elastic1DLinear` material.

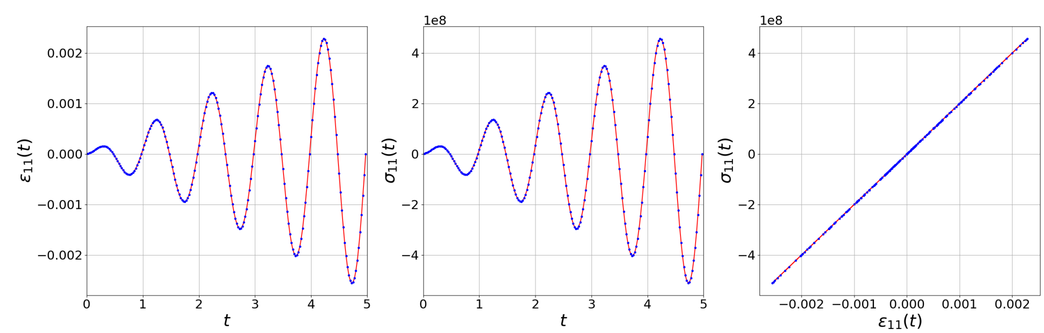


Figure 2: Nodal responses at node (2): Analytical (...), SeismoVLAB (——).

The root mean square error for the strain is : `1.67463e-12`, while The maximum absolute difference for the strain is : `4.88190e-12`.

DEBUG CASE : A02-DY_Lin_2D_Viscous_ZeroLength

Problem setting is shown in Figure 3 and is defined to test `ZeroLength1D` element with material type `Viscous1DLinear`. The material has a viscous coefficient $\eta = 0.125\ Pa\cdot s$. The nodes (1), and (2) have the coordinate $(x, y) = (0.0, 0.0)$. Node (1) is fixed in X- and Y-directions, while node (2) is fixed in Y-direction. For dynamic analysis, the nodal stress applied at node (2) is defined as $\sigma(t) = 0.1075 \cdot t\ \sin(2\pi\, t)\ Pa$. The responses are verified against analytical solution. Figure 127 shows uniaxial strain, stress, and material constitutive responses at node (2).

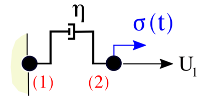


Figure 3: Varification for `ZeroLength1D` with `Viscous1DLinear` material.

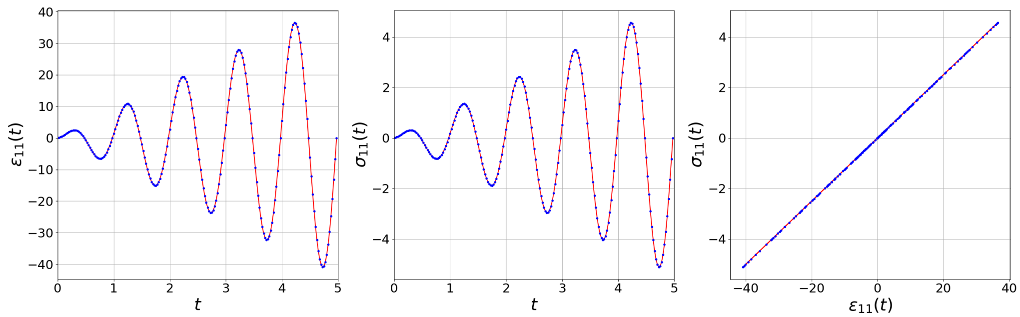


Figure 4: Nodal responses at node (2): Analytical (. . .), SeismoVLAB (——).

The root mean square error for the strain is : `2.06078e-08`, while The maximum absolute difference for the strain is : `4.98195e-08`.

DEBUG CASE : A03-DY_Lin_2D_Plastic_ZeroLength

Problem setting is shown in Figure 86 and is defined to test `Plastic1DJ2` element with material type `Plastic1DJ2`. The material has modulus of elasticity $E = 200\ GPa$, and a Poisson's ratio $\nu = 0.25$, hardening modulus $H = 375$, and kinematic modulus $K = 0.0$, the yield stress is taken as $\sigma_Y = 250$. The nodes (1), and (2) have the coordinate $(x, y) = (0.0, 0.0)$. Node (1) is fixed in X- and Y-directions, while node (2) is fixed in Y-direction. For dynamic analysis, the nodal stress applied at node (2) is defined as $\sigma(t) = 107.5 \cdot t\ \sin(2\pi\, t)\ Pa$. The responses are verified against analytical solution. Figure 6 shows uniaxial strain, stress, and material constitutive response at node (2).

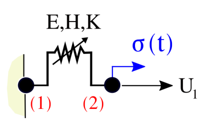


Figure 5: Varification for `ZeroLength1D` with `Plastic1DJ2` material.

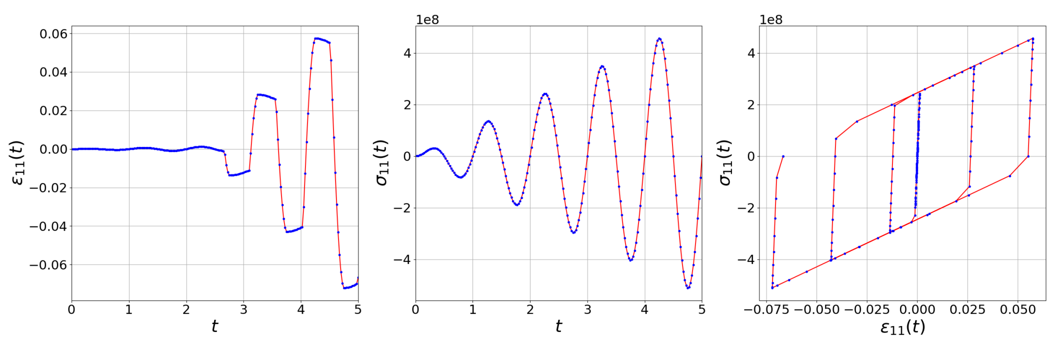


Figure 6: Nodal responses at node (2): Analytical (. . .), SeismoVLAB (——).

The root mean square error for the strain is : `1.81191e-08`, while The maximum absolute difference for the strain is : `4.95000e-08`.

DEBUG CASE : A04-DY_Lin_1DPointMass_Elastic_ZeroLength

Problem setting is shown in Figure 7 and correspond to a mass-spring-dashpot oscillator for which $M = 1$ $[kg]$, $K = 4$ $[N/m]$, $C = 0.2$ $[N\,s/m]$. This problem tests `Point Mass` and `ZeroLength1D`. The nodes (1), and (2) have the coordinate $(x, y) = (0.0, 0.0)$. Node (1) is fixed in X- and Y-directions, while node (2) is fixed in Y-direction. For dynamic analysis, the nodal force applied at node (2) is defined as $P(t) = 1.000 \sin(\omega_n\, t)$ N, with $\omega_n = 4$ $[rad/s]$. The responses are verified against analytical solution. Figure 8 shows the displacement and reactive force responses at node (2).

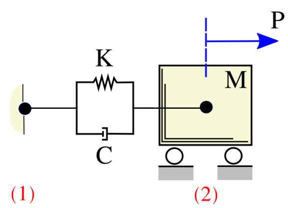


Figure 7: Varification for `ZeroLength1D` with `Viscous1DLinear` material.

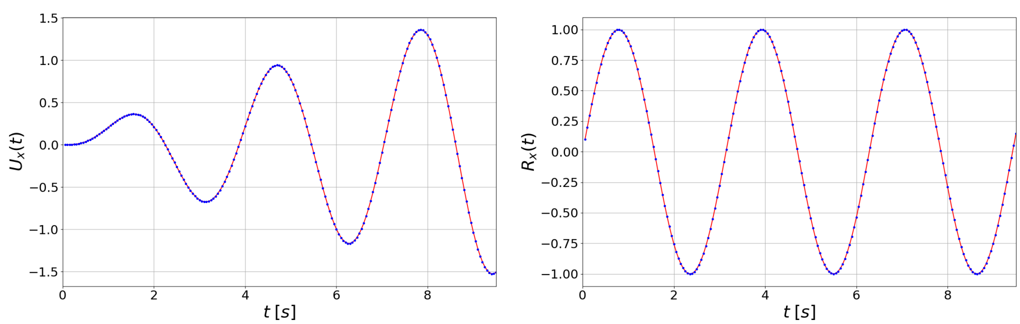


Figure 8: Nodal responses at node (2): Analytical (...), SeismoVLAB (——).

The root mean square error for the displacements and reaction are : (`0.00379617`, `3.63795e-09`), while the maximum relative error for the displacement and reaction are : (`0.00940595`,`9.19944e-09`) respectively.

DEBUG CASE : A05-DY_Lin_Hertzian_Contact_ZeroLength

Problem setting is shown in Figure 9 and correspond to a Hertzian contact oscillator for which $M = 0.0035$ $[kg]$, $k_1 = 4$ $[N/m]$, $k_2 = 4$ $[N/m]$, $k_3 = 4$ $[N/m]$, and $C = 0.2$ $[N\,s/m]$. This problem tests `Hertzian1DLinear` and `ZeroLength1D`. The nodes (1), and (2) have the coordinate $(x, y) = (0.0, 0.0)$. Node (1) is fixed in X- and Y-directions, while node (2) is fixed in Y-direction. For dynamic analysis, the nodal force applied at node (2) is defined as $P(t) = 1.000 \cos(\omega\, t)$ N, with $\omega = 6.2832 \cdot 10^4$ $[rad/s]$. The responses are verified against analytical solution. Figure 10 shows the displacement and reactive force responses at node (2).

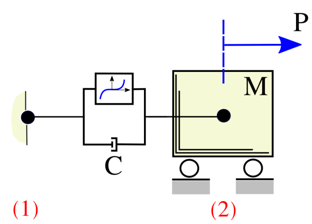


Figure 9: Varification for `ZeroLength1D` with `Hertzian1DLinear` material.

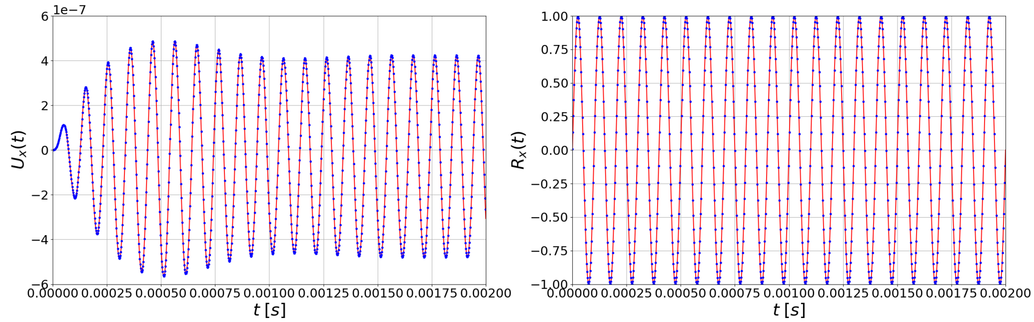


Figure 10: Nodal responses at node (2): Analytical (. . .), SeismoVLAB (——).

The root mean square error for the displacements and reaction are : (`6.81715e-10`, `8.12188e-09`), while the maximum relative error for the displacement and reaction are : (`1.72854e-09`,`2.28790e-08`) respectively.

DEBUG CASE : A06-DY_Lin_1DPointMass_Elastic_ZeroLength_SupportMotion

Problem setting is shown in Figure 11 and correspond to a mass-spring-dashpot oscillator for which $M = 1$ $[kg]$, $K = 4$ $[N/m]$, $C = 0.2$ $[N\,s/m]$. This problem tests `Point Mass` and `ZeroLength1D`. The nodes (1), and (2) have the coordinate $(x, y) = (0.0, 0.0)$. Node (1) is fixed in X- and Y-directions, while node (2) is fixed in Y-direction. For dynamic analysis, a support motion is applied at node (1) is defined as $u_g(t) = 1.000\,\sin(\omega_n\,t)\;m$, with $\omega_n = 2$ $[rad/s]$. The responses are verified against analytical solution. Figure 12 shows the displacement and reactive force responses at node (2).

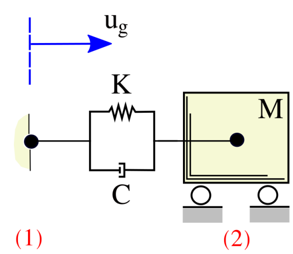


Figure 11: Varification for `ZeroLength1D` with `Viscous1DLinear` material.

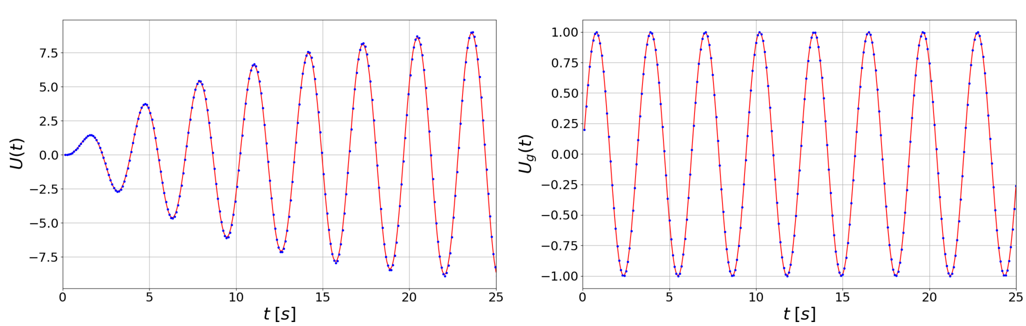


Figure 12: Nodal responses at node (2): Analytical (. . .), SeismoVLAB (——).

DEBUG CASE : A08-DY_2D_UniAxial_BoucWen_Link_Param1

The problem showed in Figure 13 is defined to test `UnxBoucWen2DLink` element and its behavior is defined on local axis 1. The link properties are $\alpha = 1.0$, $\mu = 2.0$, $\beta = 0.5$, $\gamma = 0.5$, $\eta = 1.0$, $f_y = 250.0$, $k_0 = 250.0$, $\alpha_1 = 0.1$, and $\alpha_2 = 0.0$. Nodes (1) has coordinate $(x, y) = (0.0, 0.0)$ and (2) has coordinate $(x, y) = (0.5, 0.0)$. Node (1) is fixed in X and Y directions, while node (2) is fixed in Y direction. For dynamic analysis, a point load is applied at node

(2). Responses are verified against analytical solution. Figure 14 shows the strain, stress, and material constitutive responses at (2).

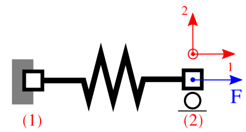


Figure 13: Varification for `UnxBoucWen2DLink` element.

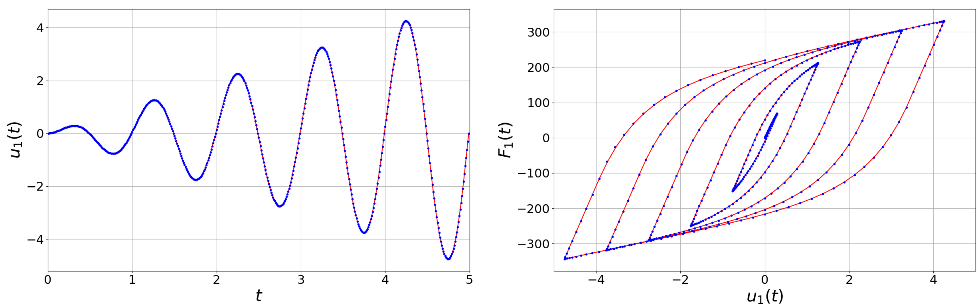


Figure 14: Nodal responses at node (2): Analytical (. . .), SeismoVLAB (——).

DEBUG CASE : A10-DY_3D_UniAxial_BoucWen_Link_Param1

The problem showed in Figure 15 is defined to test `UnxBoucWen3DLink` element and its behavior is defined on local axis 2. The link properties are $\alpha = 1.0$, $\mu = 2.0$, $\beta = 0.5$, $\gamma = 0.5$, $\eta = 1.0$, $f_y = 250.0$, $k_0 = 250.0$, $\alpha_1 = 0.1$, and $\alpha_2 = 0.0$. Nodes (1) has coordinate $(x, y, z) = (0.0, 0.0, 0.0)$ and (2) has coordinate $(x, y, z) = (0.0, 0.0, 0.5)$. Node (1) is fixed in X, Y and Z directions, while node (2) is fixed in Y and Z direction. For dynamic analysis, a point load is applied at node (2). Responses are verified against analytical solution. Figure 16 shows the strain, stress, and material constitutive responses at (2).

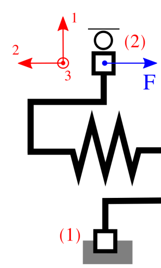


Figure 15: Varification for `UnxBoucWen3DLink` element.

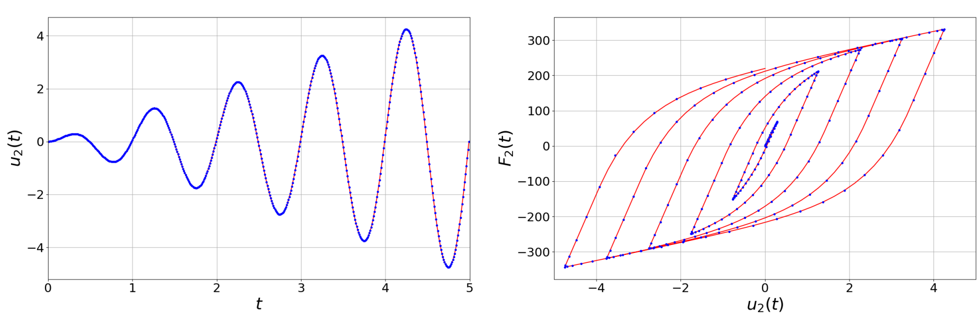


Figure 16: Nodal responses at node (2): Analytical (. . .), SeismoVLAB (——).

DEBUG CASE : A12-DY_2D_UniAxial_YamamotoHDRB_Link

The problem showed in Figure 17 is defined to test `HDRBYamamoto2DLink` element and its behavior is defined on local axis 2 and 3. The link properties are $D_e = 1.3$, $D_i = 0.3$, and $H_r = 0.261$. Nodes (1) has coordinate $(x, y) = (0.0, 0.0)$ and (2) has coordinate $(x, y) = (0.0, 0.5)$. Node (1) is fixed in X and Y directions, while node (2) is fixed in Y direction. For dynamic analysis, a point load is applied at node (2). Responses are verified against analytical solution. Figure 18 shows the strain, stress, and material constitutive responses at (2).

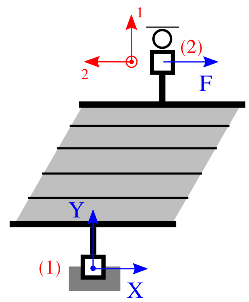


Figure 17: Varification for `HDRBYamamoto2DLink` element.

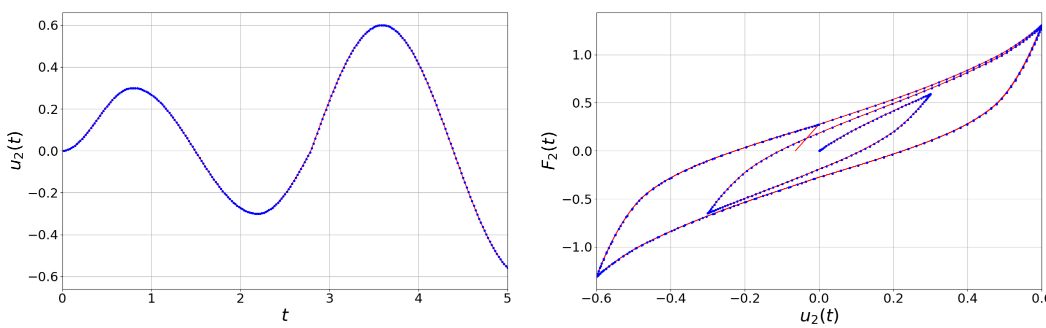


Figure 18: Nodal responses at node (2): Analytical (...), SeismoVLAB (——).

DEBUG CASE : A13-DY_3D_UniAxial_YamamotoHDRB_Link

The problem showed in Figure 19 is defined to test `HDRBYamamoto3DLink` element and its behavior is defined on local axis 2 and 3. The link properties are $D_e = 1.3$, $D_i = 0.3$, and $H_r = 0.261$. Nodes (1) has coordinate $(x, y, z) = (0.0, 0.0, 0.0)$ and (2) has coordinate $(x, y, z) = (0.0, 0.0, 0.5)$. Node (1) is fixed in X, Y and Z directions, while node (2) is fixed in Z direction. For dynamic analysis, a point load is applied at node (2). Responses are verified against analytical solution. Figure 20 shows the strain, stress, and material constitutive responses at (2).

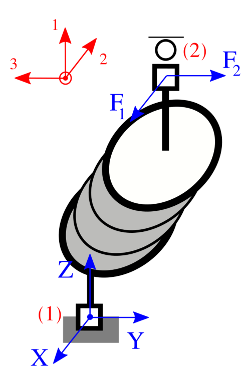


Figure 19: Varification for `HDRBYamamoto3DLink` element.

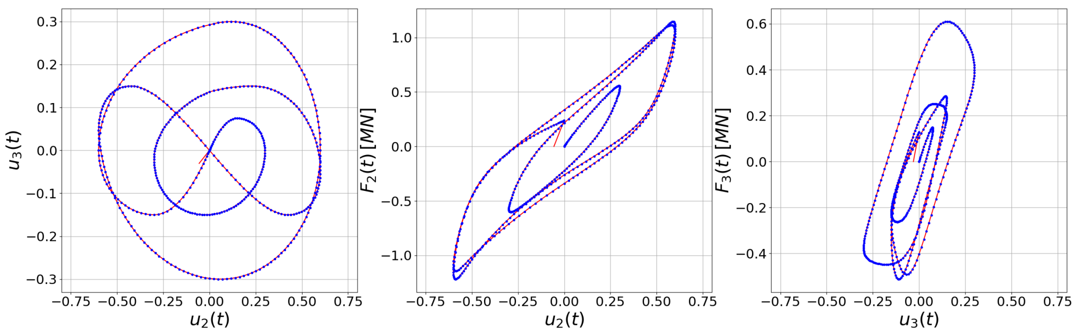


Figure 20: Nodal responses at node (2): Analytical (. . .), SeismoVLAB (——).

DEBUG CASE : A15-DY_1D_Material_Fiber_Elastic

Problem setting is shown in Figure 21 and is defined to test `ZeroLength1D` element with material type `Elastic1DFiber`. The material has modulus of elasticity $E = 29000.0$ and Poisson's ratio $\nu = 0.0$. The nodes (1), and (2) have the coordinate $(x, y) = (0.0, 0.0)$. Node (1) and (2) are fixed in X- and Y-directions, but a horizontal support motion is prescribed at Node (2) as $\Delta(t) = 0.0013 \cdot t \sin(0.4\pi\, t)$. The responses are verified against OpenSees solution. Figure **??** shows uniaxial strain, stress, and material constitutive response at node (2).

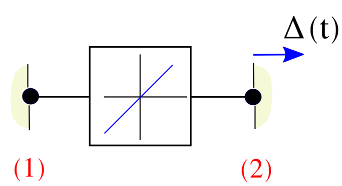


Figure 21: Varification for `ZeroLength1D` with `Elastic1DFiber` material.

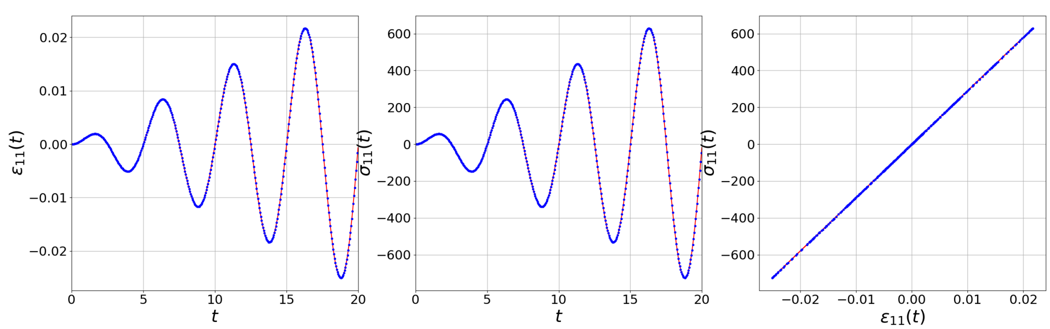


Figure 22: Nodal responses at node (2): OpenSees (. . .), SeismoVLAB (——).

The root mean square error for the strain is : `1.79036e-11`, while The maximum absolute difference for the strain is : `4.98581e-11`.

DEBUG CASE : A16-DY_1D_Material_Fiber_Elastic_Gap

Problem setting is shown in Figure 23 and is defined to test `ZeroLength1D` element with material type `Elastic1DGap`. The material has modulus of elasticity $E = 100.0$ and a gap of $\epsilon_g = 0.0025$. The nodes (1), and (2) have the coordinate $(x, y) = (0.0, 0.0)$. Node (1) and (2) are fixed in X- and Y-directions, but a horizontal support motion is prescribed at Node (2) as $\Delta(t) = 0.0013 \cdot t \sin(0.4\pi\, t)$. The responses are verified against analytical solution. Figure **??** shows uniaxial strain, stress, and material constitutive response at node (2).

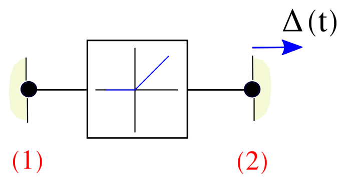


Figure 23: Varification for `ZeroLength1D` with `Elastic1DGap` material.

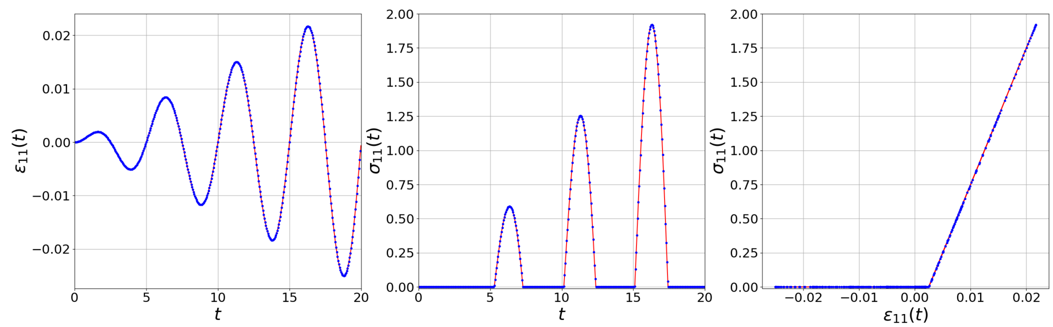


Figure 24: Nodal responses at node (2): OpenSees (...), SeismoVLAB (——).

The root mean square error for the strain is : `1.79036e-11`, while The maximum absolute difference for the strain is : `4.98581e-11`.

DEBUG CASE : A17-DY_1D_Material_Fiber_Concrete

Problem setting is shown in Figure 25 and is defined to test `ZeroLength1D` element with material type `Concrete1DFiber`. The material has compression strength $f_c = -5.73$, $e_{cc} = -0.003$, crusing strength $f_{cu} = -1.146$, $e_{cu} = -0.021$, $\lambda = 0.05$, tensile strength $ft = 0.28$, elasticity modulus $Et = 430.0$. The nodes (1), and (2) have the coordinate $(x, y) = (0.0, 0.0)$. Node (1) and (2) are fixed in X- and Y-directions, but a horizontal support motion is prescribed at Node (2) as $\Delta(t) = 0.0013 \cdot t\ \sin(0.4\pi\, t)$. The responses are verified against OpenSees solution. Figure 26 shows uniaxial strain, stress, and material constitutive response at node (2).

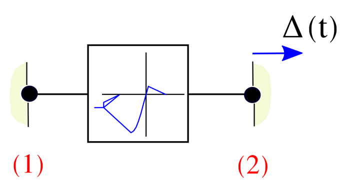


Figure 25: Varification for `ZeroLength1D` with `Concrete1DFiber` material.

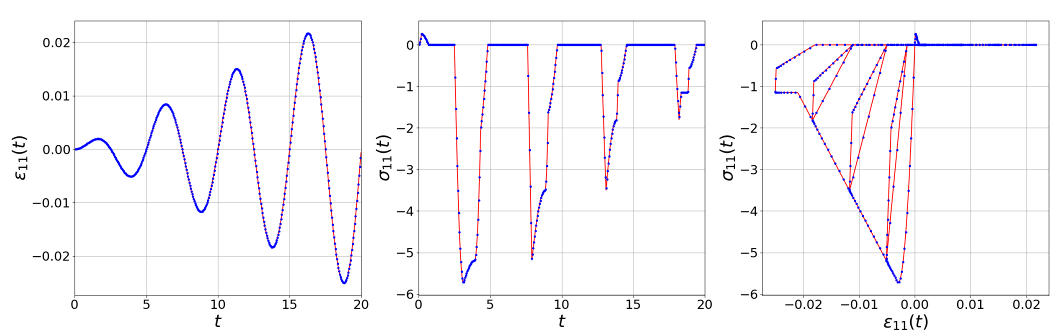


Figure 26: Nodal responses at node (2): OpenSees (...), SeismoVLAB (——).

The root mean square error for the strain is : `1.79036e-11`, while The maximum absolute difference for the strain is : `4.98581e-11`.

DEBUG CASE : A18-DY_1D_Material_Fiber_Steel

Problem setting is shown in Figure 27 and is defined to test `ZeroLength1D` element with material type `Steel1DFiber`. The material has modulus of elasticity $E = 29000.0$, and yield stress $fy = 60.0$, parameters $b = 0.01$, $R0 = 20.0$, $cR1 = 0.925$, $cR2 = 0.15$, $a1 = a3 = 0.0$, and $a2 = a4 = 1.0$. The nodes (1), and (2) have the coordinate $(x, y) = (0.0, 0.0)$. Node (1) and (2) are fixed in X- and Y-directions, but a horizontal support motion is prescribed at Node (2) as $\Delta(t) = 0.0013 \cdot t \sin(0.4\pi\, t)$. The responses are verified against OpenSees solution. Figure 28 shows uniaxial strain, stress, and material constitutive response at node (2).

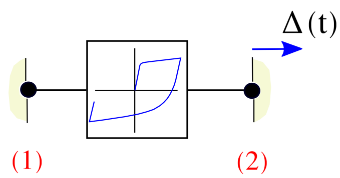


Figure 27: Varification for `ZeroLength1D` with `Steel1DFiber` material.

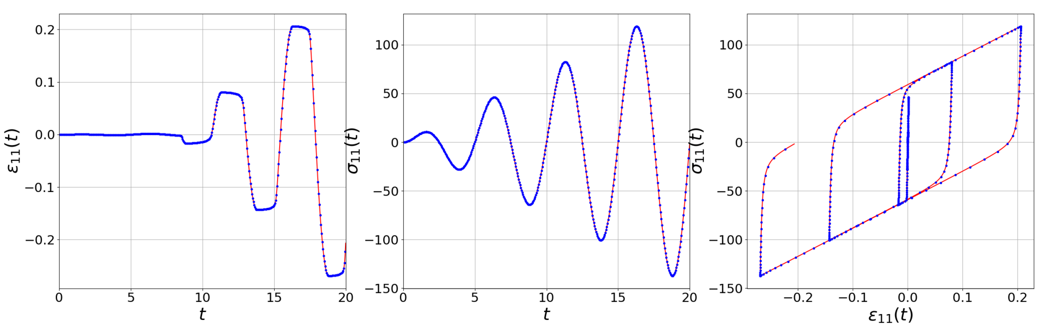


Figure 28: Nodal responses at node (2): OpenSees (...), SeismoVLAB (——).

The root mean square error for the strain is : `0.00000`, while The maximum absolute difference for the strain is : `0.00000`.

DEBUG CASE : A19-DY_1D_Material_Fiber_Plastic_Gap

Problem setting is shown in Figure 29 and is defined to test `ZeroLength1D` element with material type `Plastic1DGap`. The material has tension strength $fy = 10.0$, modulus of elasticity $E = 50.0$, and a gap of $\epsilon_g = 0.2$. The nodes (1), and (2) have the coordinate $(x, y) = (0.0, 0.0)$. Node (1) and (2) are fixed in X- and Y-directions, but a horizontal support motion is prescribed at Node (2) as in Figure 30. The responses are verified against OpenSees solution. Figure 30 shows uniaxial strain, stress, and material constitutive response at node (2).

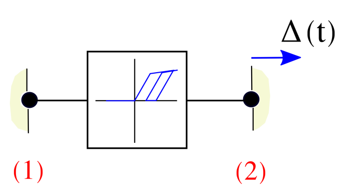


Figure 29: Varification for `ZeroLength1D` with `Plastic1DGap` material.

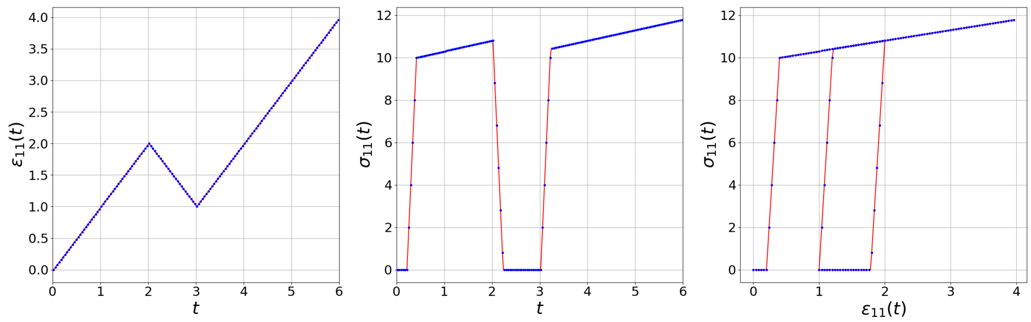


Figure 30: Nodal responses at node (2): OpenSees (...), SeismoVLAB (——).

The root mean square error for the strain is : `0.00000`, while The maximum absolute difference for the strain is : `0.00000`.

DEBUG CASE : C01-ST_Lin_2DAxial_Elastic_Truss2

Problem setting is shown in Figure 31 and is defined to test `lin2DTruss2` element with material type `Elastic1DLinear`. The material has modulus of elasticity $E = 68.9\ GPa$. The nodes (1), and (2) have the coordinates $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1) is fixed in X-, and Y-directions, while node (2) is fixed in Y-direction. The truss element has an area of $A = 0.0025\ m^2$, and a length of $L = 1.00\ m$. For static analysis, the nodal force applied at node (2) is defined as $P = 1000\ N$. The responses are verified against analytical solution.

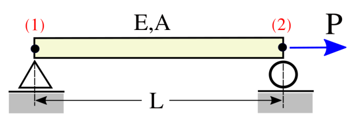


Figure 31: Varification for `lin2DTruss2` with `Elastic1DLinear` material.

In this case, the relative absolute error for the displacement is : `9.00000e-11`, while The relative absolute error for the strain and stress are : `1.00000e-08`, `6.98000e-09` repectively.

DEBUG CASE : C02-ST_Lin_2DRoof_Elastic_Truss2

The problem showed in Figure 32 is a roof defined to test `lin2DTruss2` element with material type `Elastic1DLinear` and the local axis transformations for Truss elements. The material has a elasticity moduli $E = 250\ GPa$. Nodes (1), (2), (4), and (5) have coordinate $(0.0, 0.0)$, $(3.0, 0.0)$, $(9.0, 0.0)$, and $(3.0, 4.0)$ respectively. Node (4) is fixed in X, and Y directions, while node (2) is fixed in Y direction. The truss elements have an area $A = 300 \cdot 10^{-}6\, m^2$. A vertical load is placed at node (1) has magnitude $P = 200\ kN$. Responses are verified against analytical solution.

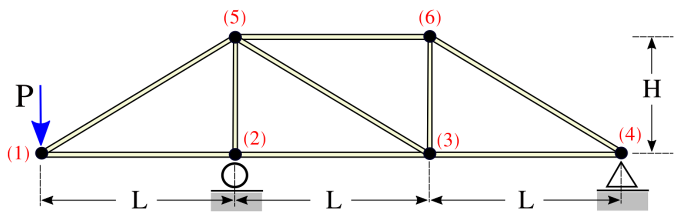


Figure 32: Varification for `lin2DTruss2` with `Elastic1DLinear` material.

The relative error for the vertical deformation at node (1) and (5) are : `(4.33735e-09, 0.00000)`, while the maximum relative error for the internal axial forces for all elements is : `0.00000`.

DEBUG CASE : C03-ST_Lin_3DAxial_Elastic_Truss2

Problem setting is shown in Figure 33 and is defined to test `lin3DTruss2` element with material type `Elastic1DLinear`. The material has modulus of elasticity $E = 68.9\ GPa$, and a Poisson's ratio $\nu = 0.33$. The nodes (1), and (2) have the coordinate $(0.0, 0.0, 0.0)$ and $(1.0, 0.0, 0.0)$ respectively. Node (1) is fixed in X-, Y-, and Z-directions, while node (2) is fixed in Y-, and Z-direction. The truss element has an area of $1.00\ m^2$, and a length of $L = 1.00\ m$. For static analysis, the nodal force applied at node (2) is defined as $P = 1000\ N$. The responses are verified against analytical solution.

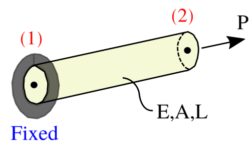


Figure 33: Varification for `lin3DTruss2` with `Elastic1DLinear` material.

In this case, the relative absolute error for the displacement is : `9.00000e-11`, while The relative absolute error for the strain and stress are : `1.00000e-08`, `6.98000e-09` repectively.

DEBUG CASE : C04-ST_Lin_3DAxial_Plastic_Truss2

Problem setting is shown in Figure 34 and defined to test `lin3DTruss2` element with material type `Plastic1DJ2`. The material has elasticity modulus $E = 1.00\ Pa$, and a Poisson's ratio $\nu = 0.25$, hardening modulus $H = 0.25\ Pa$, and kinematic modulus $K = 0.25\ Pa$, the yield stress is taken as $\sigma_Y = 5.0\ Pa$. The nodes (1), and (2) have the coordinate $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1) is fixed in X-, Y-, and Z-directions, while node (2) is fixed in Y-, and Z-direction. The truss has an area of $1.00\ m^2$, and a length of $L = 1.00\ m$. For static analysis, the nodal force applied at node (2) is defined as $P = 1\ kN$. The responses are verified against analytical solution.

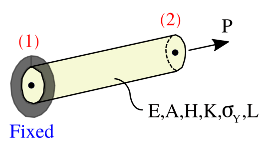


Figure 34: Varification for `lin3DTruss2` with `Plastic1DJ2` material.

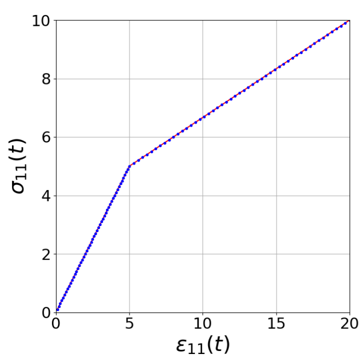


Figure 35: Nodal responses at node (2): Analytical (...), SeismoVLAB (——).

The maximum absolute error for the strain and stress are : `0.00000`, `1.77636e-15` respectively.

DEBUG CASE : C05-ST_Lin_3DPiramid_Elastic_Truss2

The problem showed in Figure 36 is an piramid defined to test `lin3DTruss2` element with material type `Elastic1DLinear` and the local axis transformations for Truss elements. The material has a elasticity moduli $E = 68.9\ GPa$, and a

Poisson's ratio $\nu = 0.33$. Nodes (1), (2), (3), and (4) have coordinate $(-1.0, -4.0, 0.0)$, $(2.0, 0.0, 0.0)$, $(-1.0, 4.0, 0.0)$, and $(0.0, 0.0, 8.0)$ respectively. Node (1) is fixed in X, Y, and Z directions, while node (2) and (3) are fixed in Z-direction. The truss elements have an area $A = 0.0025\, m^2$. A vertical load is placed at node (4) with magnitude $P = 200 \cdot 10^3\ N$. Responses are verified against analytical solution.

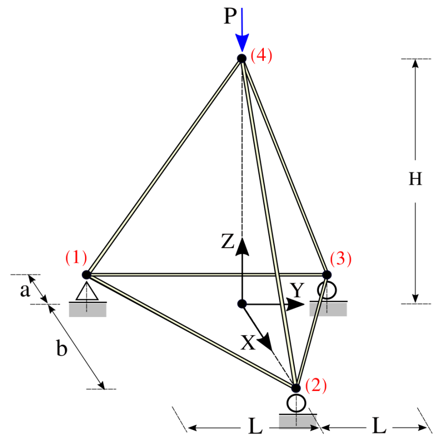


Figure 36: Varification for `lin3DTruss2` with `Elastic1DLinear` material.

The relative error for the vertical deformation at node (4) is : `9.40308e-06`, while the maximum relative error for the internal axial forces for all elements is : `5.88235e-10`.

DEBUG CASE : C08-ST_kin_2DCantilever_Elastic_Truss2

Problem setting is shown in Figure 37 and is defined to test `kin2DTruss2` element with material type `Elastic1DLinear`. For this example, all truss members have a cross-sectional area, $A = 0.1\,[in^2]$, and modulus of elasticity, $E = 29000\,[ksi]$. The truss is 10 inches long, and horizontal and vertical members are 0.5 inch long. As a result of the above dimensions there are 42 nodes and 81 members. A vertical load of $20\,[kips]$ is placed at Node (42) The tolerance used for equilibrium iterations is 10^{-6}. The responses are verified against numerical solution provied by Louie L. Yaw. Figure 38 shows the force displacement curve at node (42).

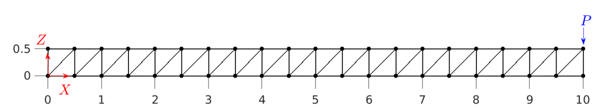


Figure 37: Varification for `kin2DTruss2` with `Elastic1DLinear` material.

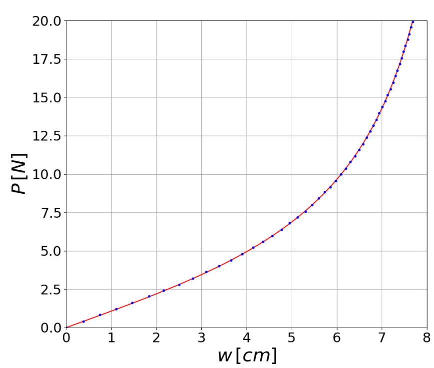


Figure 38: Nodal responses at node (2): Analytical (. . .), SeismoVLAB (——).

DEBUG CASE : C09-ST_kin_3DCantilever_Elastic_Truss2

Problem setting is shown in Figure 39 and is defined to test `kin3DTruss2` element with material type `Elastic1DLinear`. For this example, the cantilevered space truss is $10\,[m]$ long, $0.2\,[m]$ wide and $0.5\,[m]$ deep. The truss has two top chord members and two bottom chord members. For all truss members (both nodes at y=0 or both nodes at y=0.2 inches) the area used is $1.0\,[cm^2]$. All members have a modulus of elasticity of $E = 1\,[kN/cm^2]$. The nodes at the support for the cantilever are restrained in the xyz directions. Two vertical forces of magnitude $3.5\,10^{-4}[kN]$ are placed at Node (42) and (84). The tolerance used for equilibrium iterations is 10^{-6}. The responses are verified against numerical solution provied by Louie L. Yaw. Figure 40 shows the force displacement curve at node (42).

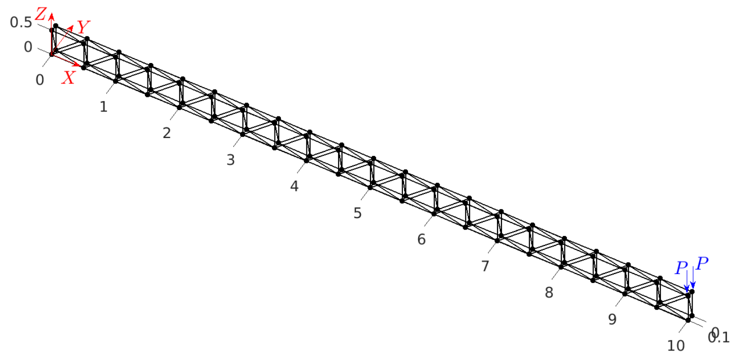


Figure 39: Varification for `kin3DTruss2` with `Elastic1DLinear` material.

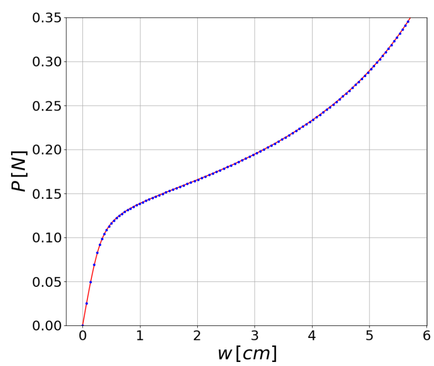


Figure 40: Force displacement curve at (42): Analytical (...), SeismoVLAB (——).

DEBUG CASE : C11-ST_Lin_2DSurface_Elastic_Truss2

Problem setting is shown in Figure 41 and is defined to test `lin2DTruss2` element with material type `Elastic1DLinear`. The material has modulus of elasticity $E = 100000\ Pa$. The nodes (1), and (6) have the coordinates $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1) is fixed in X-, and Y-directions, while nodes (2), (3), (4), (5), and (6) are fixed in Y-direction. The truss element has an area of $A = 0.04\ m^2$, and a length of $L = 1.00\ m$. For static analysis, the surface load is applied on all elements and defined as $q = 10\ N/m$. The responses are verified against analytical solution.

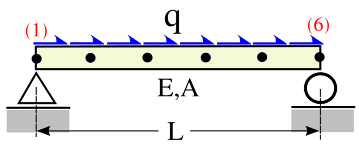


Figure 41: Varification for `lin2DTruss2` with `Elastic1DLinear` material.

In this case, the relative absolute error for the displacement is : `0.00000`, while The relative absolute error for the strain and stress are : `0.100000`, `0.100000` repectively.

DEBUG CASE : C12-ST_Lin_3DSurface_Elastic_Truss2

Problem setting is shown in Figure 42 and is defined to test `lin3DTruss2` element with material type `Elastic1DLinear`. The material has modulus of elasticity $E = 100000\ Pa$. The nodes (1), and (6) have the coordinates $(0.0, 0.0, 0.0)$ and $(1.0, 0.0, 0.0)$ respectively. Node (1) is fixed in X-, Y- and Z-directions, while nodes (2), (3), (4), (5), and (6) are fixed in Y- and Z-directions. The truss element has an area of $A = 0.04\ m^2$, and a length of $L = 1.00\ m$. For static analysis, the surface load is applied on all elements and defined as $q = 10\ N/m$. The responses are verified against analytical solution.

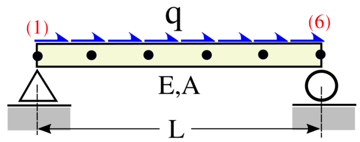


Figure 42: Varification for `lin3DTruss2` with `Elastic1DLinear` material.

In this case, the relative absolute error for the displacement is : `0.00000`, while The relative absolute error for the strain and stress are : `0.100000`, `0.100000` repectively.

DEBUG CASE : C15-ST_Lin_2DSurface_Elastic_Truss3

Problem setting is shown in Figure 43 and is defined to test `lin2DTruss3` element with material type `Elastic1DLinear`. The material has modulus of elasticity $E = 100000\ Pa$. The nodes (1), and (6) have the coordinates $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1) is fixed in X-, and Y-directions, while nodes from (2) to (11) are fixed in Y-direction. The truss element has an area of $A = 0.04\ m^2$, and a length of $L = 1.00\ m$. For static analysis, the surface load is applied on all elements and defined as $q = 10\ N/m$. The responses are verified against analytical solution.

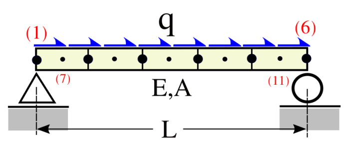


Figure 43: Varification for `lin2DTruss3` with `Elastic1DLinear` material.

In this case, the relative absolute error for the displacement is : `0.00000`, while The relative absolute error for the strain and stress are : `0.0225403`, `0.0225403` repectively.

DEBUG CASE : C16-ST_Lin_3DSurface_Elastic_Truss3

Problem setting is shown in Figure 44 and is defined to test `lin3DTruss3` element with material type `Elastic1DLinear`. The material has modulus of elasticity $E = 100000\ Pa$. The nodes (1), and (6) have the coordinates $(0.0, 0.0, 0.0)$ and $(1.0, 0.0, 0.0)$ respectively. Node (1) is fixed in X-, Y- and Z-directions, while nodes (2), (3), (4), (5), and (6) are fixed in Y- and Z-directions. The truss element has an area of $A = 0.04\ m^2$, and a length of $L = 1.00\ m$. For static analysis, the surface load is applied on all elements and defined as $q = 10\ N/m$. The responses are verified against analytical solution.

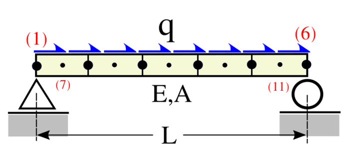


Figure 44: Varification for `lin3DTruss3` with `Elastic1DLinear` material.

In this case, the relative absolute error for the displacement is : `0.00000`, while The relative absolute error for the strain and stress are : `0.0225403`, `0.0225403` repectively.

DEBUG CASE : D01-ST_Lin_2DBernoulli_Elastic_Frame2

The problem showed in Figure 45 is defined to test `lin2DFrame2` element with material type `Elastic1DLinear`. The material has a elasticity moduli $E = 68.9\ GPa$, and a Poisson's ratio $\nu = 0.33$. Nodes (1) and node (2) have coordinate $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1) is fixed in X and Y directions, while node (2) is free. The Bernoulli beam has a rectangular cross section with $h = b = 0.05\ m$. A vertical load is placed at node (2) with magnitude $P = 1000\ N$. Responses are verified against analytical solution.

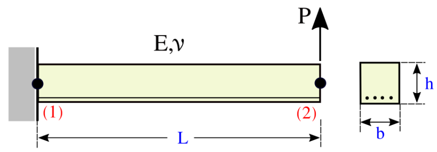


Figure 45: Varification for `lin2DFrame2` with `Elastic1DLinear` material.

The relative error for the vertical deformation is : `3.40625e-10`, while the maximum relative error for the reaction forces are : `0.00000`.

DEBUG CASE : D02-ST_Lin_2DTimoshenko_Elastic_Frame2

The problem showed in Figure 46 is defined to test `lin2DFrame2` element with material type `Elastic1DLinear`. The material has a elasticity moduli $E = 68.9\ GPa$, and a Poisson's ratio $\nu = 0.33$. Nodes (1) and node (2) have coordinate $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1) is fixed in X and Y directions, while node (2) is free. The Timoshenko beam has a rectangular cross section with $h = 0.20\ m$ and $b = 0.05\ m$. A vertical load is placed at node (2) with magnitude $P = 1000\ N$. Responses are verified against analytical solution.

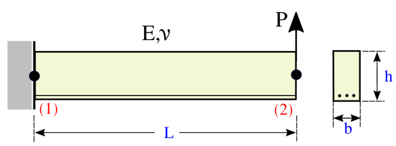


Figure 46: Varification for `lin2DFrame2` with `Elastic1DLinear` material.

The relative error for the vertical deformation is : `0.000639993`, while the maximum relative error for the reaction forces are : `0.00000`.

DEBUG CASE : D03-ST_Lin_3DBernoulli_Elastic_Frame2

The problem showed in Figure 47 is defined to test `lin2DFrame2` element with material type `Elastic1DLinear`. The material has a elasticity moduli $E = 25.0\ GPa$, and a Poisson's ratio $\nu = 0.25$. Nodes (1) and node (2) have coordinate $(0.0, 0.0, 0.0)$ and $(7.5, 0.0, 0.0)$ respectively. Node (1) is fixed in X and Y directions, while node (2) is free. The Bernoulli beam has a rectangular cross section with $h = 0.75\ m$ and $b = 0.40\ m$. A load is placed at node (2) with magnitude $P = 1\ kN$ and direction $\hat{n} = (1, 1, -1)$. Responses are verified against analytical solution.

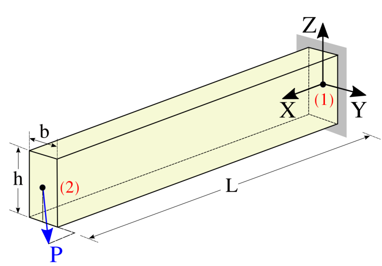


Figure 47: Varification for `lin3DFrame2` with `Elastic1DLinear` material.

The relative error for the vertical deformation is : `4.66250e-07`, while the maximum relative error for the reaction forces are : `0.00000`.

DEBUG CASE : D04-ST_Lin_3DTimoshenko_Elastic_Frame2

The problem showed in Figure 48 is defined to test `lin2DFrame2` element with material type `Elastic1DLinear`. The material has a elasticity moduli $E = 25.0\ GPa$, and a Poisson's ratio $\nu = 0.25$. Nodes (1) and node (2) have coordinate $(0.0, 0.0, 0.0)$ and $(3.5, 0.0, 0.0)$ respectively. Node (1) is fixed in X and Y directions, while node (2) is free. The Timoshenko beam has a rectangular cross section with $h = 1.0\ m$ and $b = 0.4\ m$. A load is placed at node (2) with magnitude $P = 1\ kN$ and direction $\hat{n} = (1, 1, -1)$. Responses are verified against analytical solution.

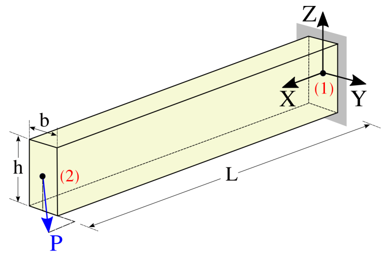


Figure 48: Varification for `lin3DFrame2` with `Elastic1DLinear` material.

The relative error for the vertical deformation is : `0.000961997`, while the maximum relative error for the reaction forces are : `0.00000`.

DEBUG CASE : D05-ST_Lin_2DBernoulliArc_Elastic_Frame2

The problem showed in Figure 49 is an arch beam defined to test `lin2DFrame2` element with material type `Elastic1DLinear` and the local axis transformations. The material has a elasticity moduli $E = 68.9\ GPa$, and a Poisson's ratio $\nu = 0.33$. Nodes (1) and node (2) have coordinate $(0.0, 10.0)$ and $(10.0, 0.0)$ respectively. Node (1) is fixed in X and Y directions, while node (20) is free. The Bernoulli beam has a rectangular cross section with $h = 1.0\ m$ and $b = 0.2\ m$. A horizontal load is placed at node (20) with magnitude $P = 1\ kN$. Responses are verified against analytical solution.

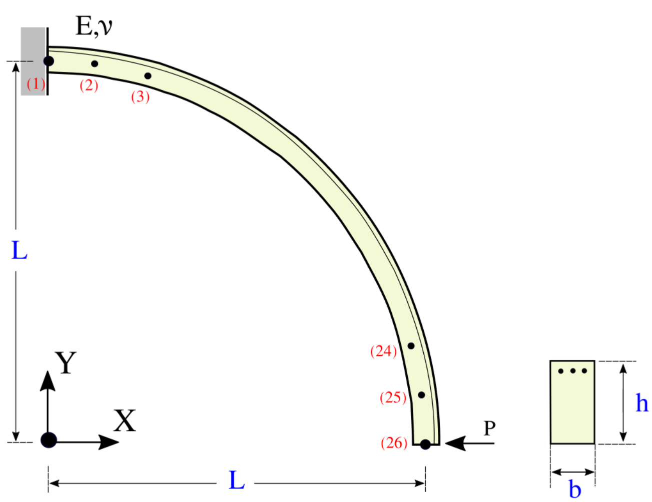


Figure 49: Varification for `lin2DFrame2` with `Elastic1DLinear` material.

The relative error for the horizontal and vertical deformation at node (20) are : `1.10623e-05`, `0.000998143` respectively. The maximum relative error for the reaction forces at node (1) is : `0.00000`.

DEBUG CASE : D06-ST_Lin_2DTimoshenkoArc_Elastic_Frame2

The problem showed in Figure 50 is an arch beam defined to test `lin2DFrame2` element with material type `Elastic1DLinear` and the local axis transformations. The material has a elasticity moduli $E = 68.9\ GPa$, and a Poisson's ratio $\nu = 0.33$. Nodes (1) and node (2) have coordinate $(0.0, 10.0)$ and $(10.0, 0.0)$ respectively. Node (1) is fixed in X and Y directions, while node (20) is free. The Timoshenko beam has a rectangular cross section with

$h = 1.0\ m$ and $b = 0.2\ m$. A horizontal load is placed at node (20) with magnitude $P = 1\ kN$. Responses are verified against approximated (and Bernoulli) analytical solution.

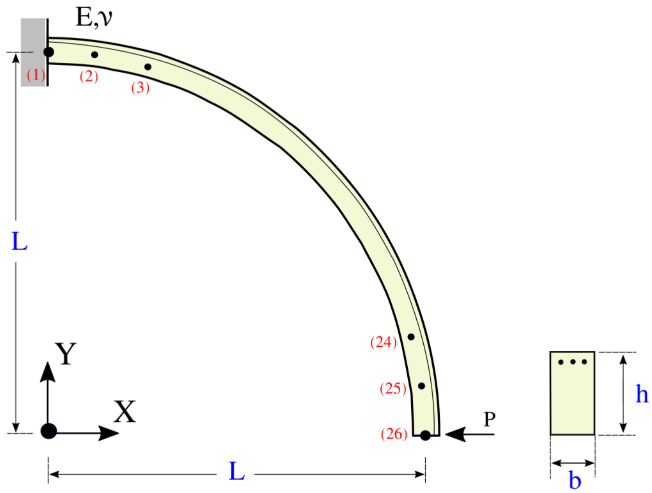


Figure 50: Verification for `lin2DFrame2` with `Elastic1DLinear` material.

The relative error for the horizontal and vertical deformation at node (20) are : `0.00267062`, `0.00166317` respectively. The maximum relative error for the reaction forces at node (1) is : `0.00000`.

DEBUG CASE : D07-ST_Lin_2DConstrainedBuilding_Elastic_Frame2

The problem showed in Figure 51 is a portal frame defined to test `lin2DFrame2` element with material type `Elastic1DLinear`, local axis transformations and kinemtic constraints. The material has a elasticity moduli $E = 2.35\ GPa$, and a Poisson's ratio $\nu = 0.20$. Nodes (1), (8) and (25) have coordinate $(0.0, 0.0)$, $(0.0, 3.5)$, and $(5.0, 0.0)$ respectively. Node (1) and (25) are fixed in X and Y directions. The Bernoulli beam is employed to model columns with rectangular cross section $h = b = 0.2\ m$ and beams with rectangular cross section $h = 1.0\ m$ and $b = 0.2\ m$. A horizontal load is placed at node (8) with magnitude $P = 1\ kN$, and kinematic constraints are enforced on the horizontal direction for all nodes from (8) to (18). Responses are verified against analytical (simplified) solution.

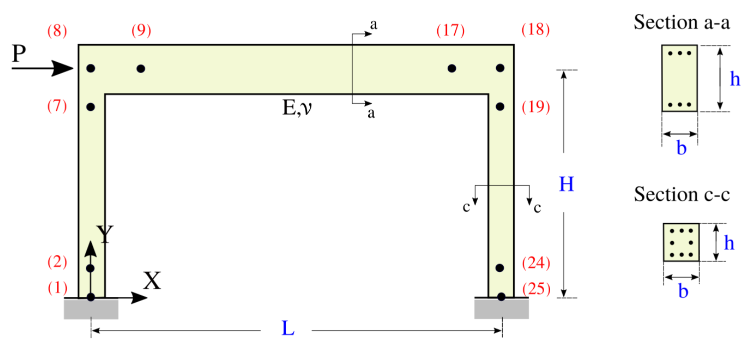


Figure 51: Varification for `lin2DFrame2` with `Elastic1DLinear` material and kinematic constraints.

The relative error for the horizontal deformation at node (8) is : `0.00727225`. The maximum relative error for the reaction forces at node (1) and (25) are : `0.00243217` and `0.00243217` repectively.

DEBUG CASE : D08-ST_Lin_2DVolForce_Elastic_Frame2

The problem showed in Figure 52 is a cantilever beam defined to test `lin2DFrame2` element with material type `Elastic1DLinear` and gravity load. The material has a elasticity modulus $E = 68.9\ GPa$, and a Poisson's ratio $\nu = 0.33$, and a density of $\rho = 2500\ kg/m^3$. Nodes (1) and node (6) have coordinate $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1) is fixed in X and Y directions, while node (6) is free. The Bernoulli beam has a rectangular cross section with $h = b = 0.05\ m$. The beam is subjected to its own weight. Responses are verified against analytical solution.

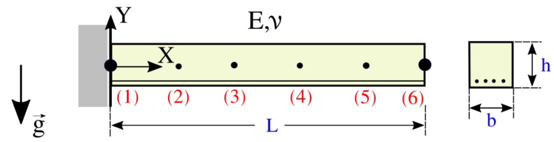


Figure 52: Varification for `lin2DFrame2` and Self-Weight Load in 2D.

The relative error for the vertical deformation at node (6) is : `1.77430e-09`. The maximum relative error for the reaction forces at node (1) is : `2.31855e-16`.

DEBUG CASE : D09-ST_Lin_3DVolForce_Elastic_Frame2

The problem showed in Figure 53 is a cantilever beam defined to test `lin3DFrame2` element with material type `Elastic1DLinear` and gravity load. The material has a elasticity modulus $E = 68.9\ GPa$, and a Poisson's ratio $\nu = 0.33$, and a density of $\rho = 2500\ kg/m^3$. Nodes (1) and node (6) have coordinate $(0.0, 0.0, 0.0)$ and $(0.0, 0.0, 1.0)$ respectively. Node (1) is fixed in X, Y, and Z directions, while node (6) is free. The Bernoulli beam has a rectangular cross section with $h = b = 0.05\ m$. The beam is subjected to its own weight. Responses are verified against analytical solution.

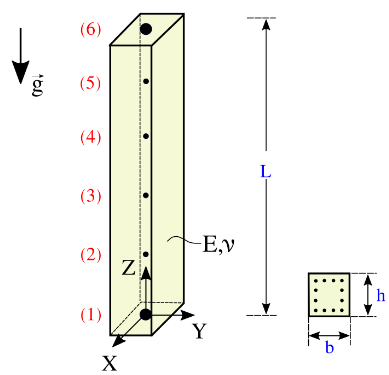


Figure 53: Varification for `lin3DFrame2` and Self-Weight Load in 3D.

The relative error for the vertical deformation at node (6) is : `1.03603e-09`. The maximum relative error for the reaction forces at node (1) is : `2.31855e-16`.

DEBUG CASE : D10-ST_kin_2DPointLoad_Bernoulli_Elastic_Frame2

Problem setting is shown in Figure 54 and is defined to test `kin2DFrame2` element with material type `Elastic1DLinear`. For this example, all beam members have a cross-sectional area, $A = 4.0\,[in^2]$, second area moment of inertia, $I = 1.3333\,[in^4]$ and modulus of elasticity, $E = 100\,[ksi]$. The beam is $10\,[in]$ long and is discretized with 10 equal length beam elements and 11 nodes. A vertical load is placed aat Node (11) of magnitude $10\,[kips]$. The tolerance used for equilibrium iterations is 10^{-3}. The responses are verified against numerical solution provied by Louie L. Yaw. Figure 55 shows the force displacement curve at node (11).

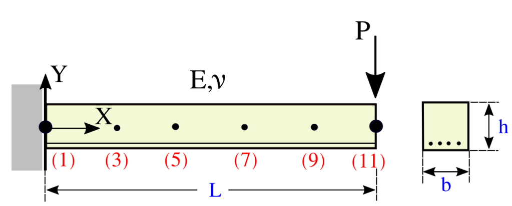


Figure 54: Varification for `kin2DFrame2` with `Elastic1DLinear` material.

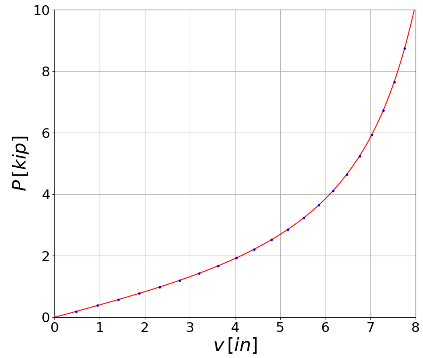


Figure 55: Force displacement curve at (11): Analytical (. . .), SeismoVLAB (——).

DEBUG CASE : D11-ST_kin_2DMomentBernoulli_Elastic_Frame2

Problem setting is shown in Figure 56 and is defined to test `kin2DFrame2` element with material type `Elastic1DLinear`. For this example, all beam members have a cross-sectional area, $A = 4.0\,[in^2]$, second area moment of inertia, $I = 1.3333\,[in^4]$ and modulus of elasticity, $E = 100\,[ksi]$. The beam is $10\,[in]$ long and is discretized with 10 equal length beam elements and 11 nodes. A moment is placed at Node (11) of magnitude $M_c = 2\pi\mathrm{EI/L}\,[kip-in]$. The tolerance used for equilibrium iterations is 10^{-3}. The responses are verified against numerical solution provied by Louie L. Yaw. Figure 57 shows the force displacement curve at node (11).

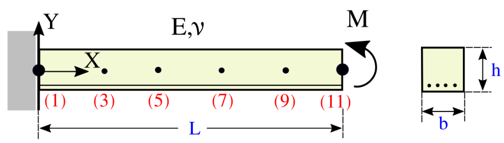


Figure 56: Varification for `kin3DTruss2` with `Elastic1DLinear` material.

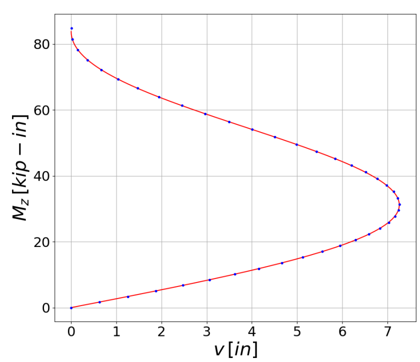


Figure 57: Moment displacement curve at (42): Analytical (. . .), SeismoVLAB (——).

DEBUG CASE : D12-ST_Lin_2DSurfaceHorizontal_Elastic_Frame2

The problem showed in Figure 58 is a cantilever beam defined to test `lin2DFrame2` element with material type `Elastic1DLinear` and surface load. The material has a elasticity modulus $E = 100000\ Pa$, and a Poisson's ratio $\nu = 0.30$. Nodes (1) and node (6) have coordinate $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1) is clamped i.e, displacement and rotation are fixed, while nodes from (2) to (6) are free. The Bernoulli beam has a rectangular cross section with $h = b = 0.20\ m$. The beam is subjected to a distributed load $q = 10\ [N/m]$. Responses are verified against analytical solution.

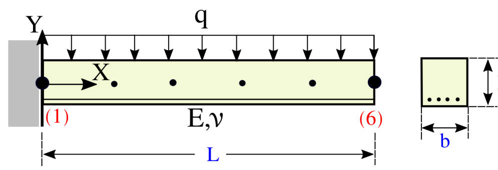


Figure 58: Varification for `lin2DFrame2` and Surface Load in 2D.

The relative error for the vertical deformation at node (6) is : `0.00000`. The maximum relative error for the reaction forces at node (1) is : `0.00000`.

DEBUG CASE : D13-ST_Lin_3DSurfaceHorizontal_Elastic_Frame2

The problem showed in Figure 59 is a cantilever beam defined to test `lin3DFrame2` element with material type `Elastic1DLinear` and surface load. The material has a elasticity modulus $E = 100000\ Pa$, and a Poisson's ratio $\nu = 0.30$. Nodes (1) and node (6) have coordinate $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1) is clamped i.e, displacement and rotation are fixed, while nodes from (2) to (6) are free. The Bernoulli beam has a rectangular cross section with $h = b = 0.20\ m$. The beam is subjected to a distributed load $q = 10\ [N/m]$. Responses are verified against analytical solution.

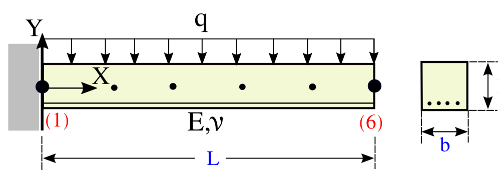


Figure 59: Varification for `lin3DFrame2` and Surface Load in 2D.

The relative error for the vertical deformation at node (6) is : `0.00000`. The maximum relative error for the reaction forces at node (1) is : `0.00000`.

DEBUG CASE : D16-DY_Free_Rectangular_3DPointLoad_Elastic_Frame2

Problem setting is shown in Figure 60 and is defined to test `Lin3Drame2` element with material type `Elastic1DLinear`. For this example, all beam members have a rectangular cross-section with height $h = 1$ and width $b = 1$, and modulus of elasticity, $E = 35000$. The beam is 10 units long and is discretized with 16 equal length beam elements and 17 nodes. A vertical dynamic load is placed aat Node (17) of constant magnitude 10. No damping is added. The responses are verified against (simplified 1 dof) analytical solution. Figure 61 shows the force displacement curve at node (17).

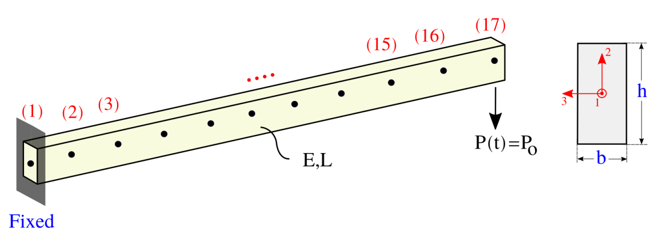


Figure 60: Varification for `lin2DFrame2` with `Lin3DRectangular` Section.

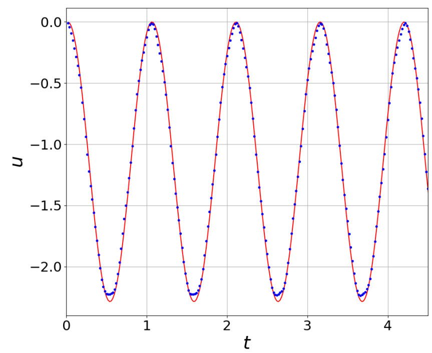


Figure 61: Displacement time series at node (17): Analytical (. . .), SeismoVLAB (——).

DEBUG CASE : D17-DY_Damped_WideFlange_3DPointLoad_Elastic_Frame2

Problem setting is shown in Figure 62 and is defined to test `Lin3Drame2` element with material type `Elastic1DLinear`. For this example, all beam members have an WideFlange cross-section with height $h = 1$, width $b = 1$, web and flange thickness $t_w = t_f = 0.1$, and modulus of elasticity, $E = 35000$. The beam is 10 units long and is discretized with 16 equal length beam elements and 17 nodes. A vertical dynamic load is placed aat Node (17) of constant magnitude 10. Rayleigh damping is added with $a_0 = 0$, and $a_1 = 0.02$. The responses are verified against analytical solution. Figure 63 shows the force displacement curve at node (17).

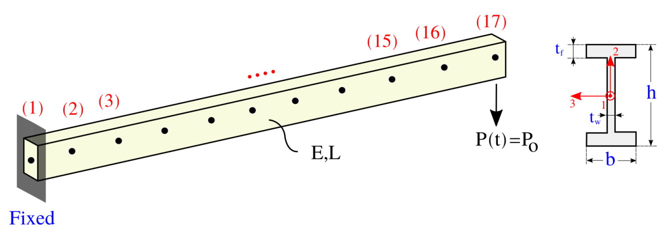


Figure 62: Varification for `lin2DFrame2` with `Lin3DWideFlange` Section.

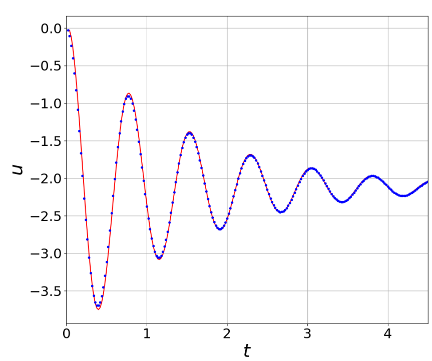


Figure 63: Displacement time series at node (17): Analytical (. . .), SeismoVLAB (——).

DEBUG CASE : D18-DY_Free_Circular_2DPointLoad_Elastic_Frame2

Problem setting is shown in Figure 64 and is defined to test `Lin2Drame2` element with material type `Elastic1DLinear`. For this example, all beam members have a circular cross-section with radius $r = 0.5$, and modulus of elasticity, $E = 35000$. The beam is 10 units long and is discretized with 16 equal length beam elements and 17 nodes. A vertical dynamic load is placed aat Node (17) of constant magnitude 10. No damping is added. The responses are verified against (simplified 1 dof) analytical solution. Figure 65 shows the force displacement curve at node (17).

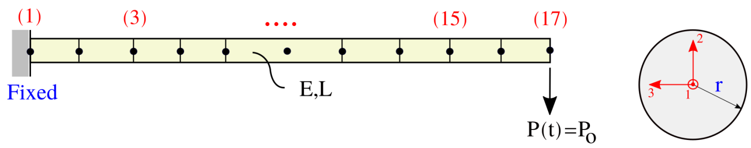


Figure 64: Varification for `lin2DFrame2` with `Lin2DCircular` Section.

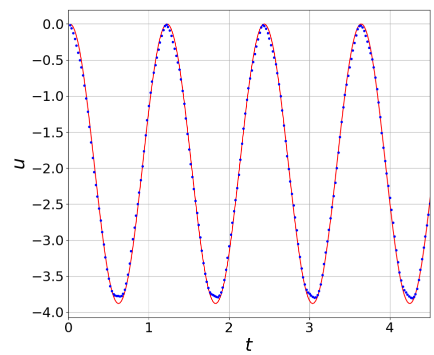


Figure 65: Displacement time series at node (17): Analytical (. . .), SeismoVLAB (——).

DEBUG CASE : D19-DY_Damped_Angle_2DPointLoad_Elastic_Frame2

Problem setting is shown in Figure 66 and is defined to test `Lin3Drame2` element with material type `Elastic1DLinear`. For this example, all beam members have an Angle cross-section with height $h = 1$, width $b = 1$, web and flange thickness $t_w = t_f = 0.1$, and modulus of elasticity, $E = 35000$. The beam is 10 units long and is discretized with 16 equal length beam elements and 17 nodes. A vertical dynamic load is placed aat Node (17) of constant magnitude 10. Rayleigh damping is added with $a_0 = 0$, and $a_1 = 0.02$. The responses are verified against analytical (simplified 1 dof) solution. Figure 67 shows the force displacement curve at node (17).

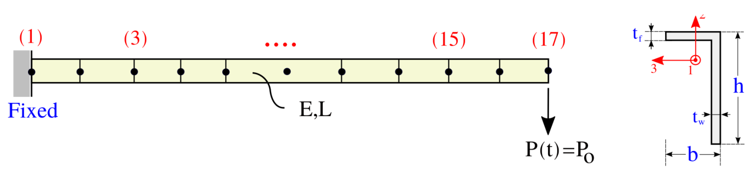


Figure 66: Varification for `lin2DFrame2` with `Lin2DAngle` Section.

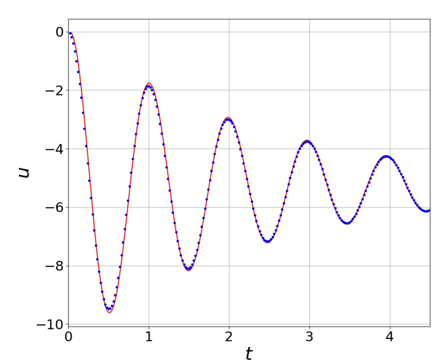


Figure 67: Displacement time series at node (17): Analytical (. . .), SeismoVLAB (——).

DEBUG CASE : D20-DY_Free_Rectangular_BodyLoad_Elastic_Frame2

Problem setting is shown in Figure 70 and is defined to test `lin3Drame2` element with material type `Elastic1DLinear`. For this example, all beam members have a rectangular cross-section with height $h = 1$ and width $b = 1$, and modulus of elasticity, $E = 20000$. The beam is 10 units long and is discretized with 16 equal length beam elements and 17 nodes. A dynamic body load is added for all elements with constant magnitude 0.1. No damping is added. The responses are verified against analytical solution. Figure 71 shows the displacement in vertical direction at node (9) as well as the shear force reaction at node (1).

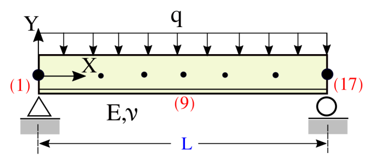


Figure 68: Varification for `lin3Drame2` with `Elastic1DLinear` material.

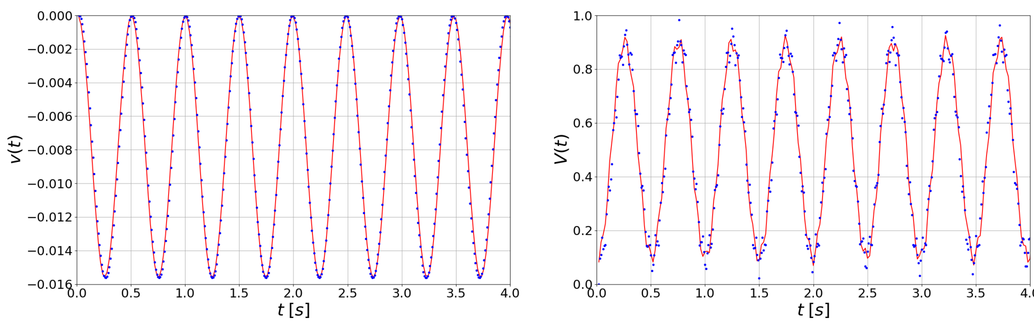


Figure 69: Displacement Nodal responses at node (9) and Shear reaction force at node (1): Analytical (. . .), SeismoVLAB (——).

DEBUG CASE : D21-DY_Damped_Rectangular_BodyLoad_Elastic_Frame2

Problem setting is shown in Figure 70 and is defined to test `lin3Drame2` element with material type `Elastic1DLinear`. For this example, all beam members have a rectangular cross-section with height $h = 1$ and width $b = 1$, and modulus of elasticity, $E = 20000$. The beam is 10 units long and is discretized with 16 equal length beam elements and 17 nodes. A dynamic body load is added for all elements with constant magnitude 0.1. Rayleigh damping is added such that $a_0 = 0.0$ and $a_1 = 0.0078$. The responses are verified against analytical solution. Figure 71 shows the displacement in vertical direction at node (9) as well as the shear force reaction at node (1).

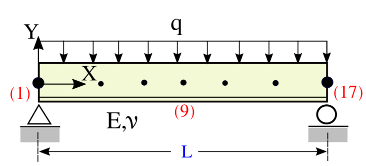


Figure 70: Varification for `lin3Drame2` with `Elastic1DLinear` material.

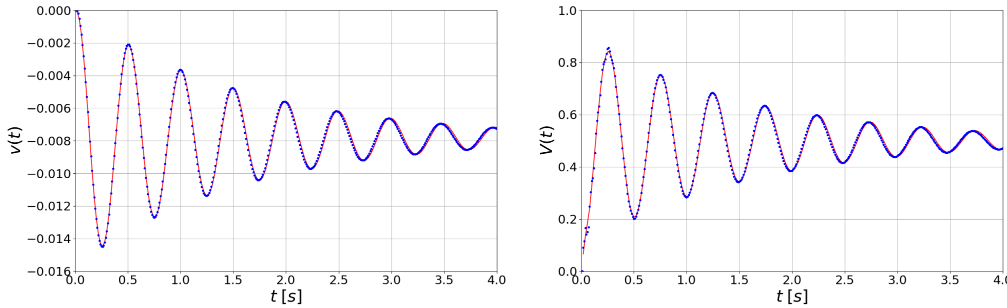


Figure 71: Displacement Nodal responses at node (5) and Shear reaction force at node (1): Analytical (...), SeismoVLAB (——).

DEBUG CASE : D22-ST_Rectangular_SupportMotion_Elastic_Frame2

Problem setting is shown in Figure 72 and is defined to test `lin3Drame2` element with material type `Elastic1DLinear`. For this example, all beam members have a rectangular cross-section with height $h = 1$ and width $b = 1$, and modulus of elasticity, $E = 20000$. The beam is 10 units long and is discretized with 16 equal length beam elements and 17 nodes. Support motion is enforced so that $\Delta = 0.5$, and $\Theta = 0.1$ at Node (1). The responses are verified against analytical solution. Figure 73 shows the displacement in vertical direction along the beam.

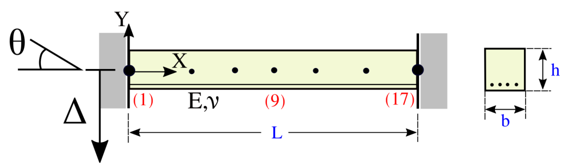


Figure 72: Varification for `lin3Drame2` with `Elastic1DLinear` material.

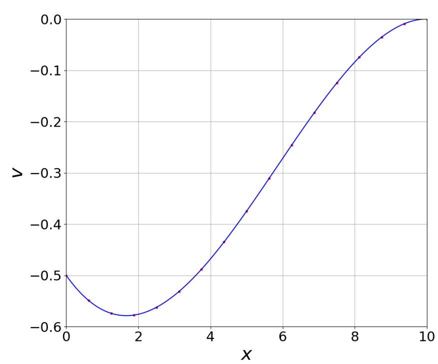


Figure 73: Nodal Displacement along the beam: Analytical (...), SeismoVLAB (——).

DEBUG CASE : D23-DY_Rectangular_SupportMotion_Elastic_Frame2

The problem showed in Figure 74 is defined to test dynamic `SupportMotion`. The problem is a beam defined to test `lin2DFrame2` element with material type `Elastic1DLinear`. The material has a elasticity modulus $E = 10$, and a Poisson's ratio $\nu = 0.25$. Nodes (1) and node (51) have coordinate $(0.0, 0.0)$ and $(10.0, 0.0)$ respectively. Node (1) and (51) are pinned i.e, displacement are fixed. The Bernoulli beam has a rectangular cross section with $h = b = 0.60$ m. Responses are verified against OpenSEES solution. Figure 75 shows the deformed configuration at different time steps.

Figure 74: Varification for `SuportMotion` constraints.

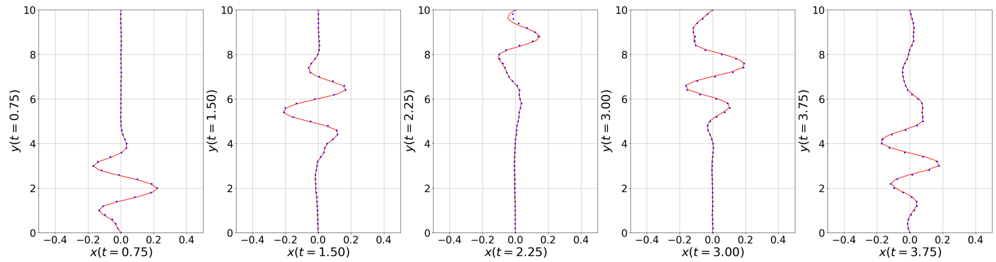


Figure 75: Deformed configuration at different time steps: Analytical (. . .), SeismoVLAB (——).

DEBUG CASE : D24-DY_WideFlange_Linear_Fiber_Section_Frame2

Problem setting is shown in Figure 76 and is defined to test `Elastic1DFiber` element with section type `Fib3DLineSection`. The material has modulus of elasticity $E = 29000000$, and a Poisson's ratio $\nu = 0.33$. The nodes (1), and (8) have the coordinate $(x, y) = (0.0, 0.0)$ and $(x, y) = (100.0, 0.0)$ respectively. Node (1) is clamped, this is fixed in X, Y-directions and rotation. For dynamic analysis, the nodal force is applied at node (8) and is defined as $\sigma(t) = 8.75 \cdot 10^5 t \,\sin(0.4\pi\, t)$. The responses are verified against OpenSees using linear elastic beam with equivalent cross section parameters. Figure 77 shows teh vertical displacement at node (2) and the moment and shear forces developed at the fixed boundary.

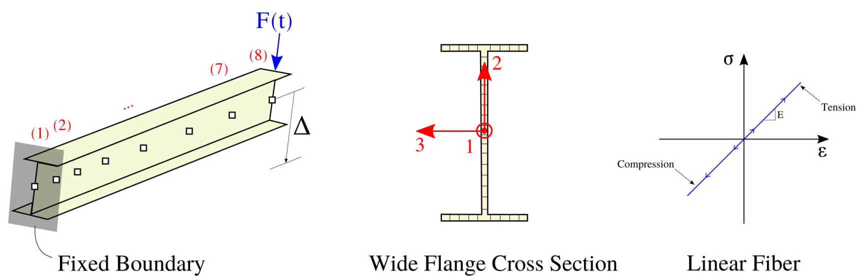


Figure 76: Varification for `Fib3DLineSection` section with `Elastic1DFiber` fiber.

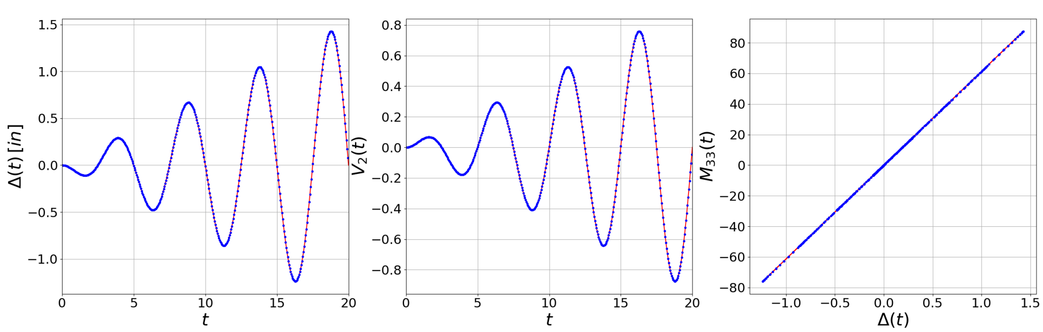


Figure 77: Nodal vertical responses at node (8) and shear force at node (8) : OpenSees (. . .), SeismoVLAB (——).

The root mean square error for the displacement is : `1.19160e-06`, while The maximum absolute difference for the displacement is : `5.83000e-06`.

DEBUG CASE : D25-DY_WideFlange_NonLinear_Fiber_Section_Frame2

Problem setting is shown in Figure 78 and is defined to test `Steel1DFiber` element with section type `Fib3DLineSection`. The material has modulus of elasticity $E = 29000000$, yield strength $fy = 60000$, $b = 0.01$, $a_1 = a_3 = 0.0$, $a_2 = a_4 = 1.0$, $cR_1 = 0.925$, $cR_2 = 0.15$, and a Poisson's ratio $\nu = 0.33$. The nodes (1), and (8) have the coordinate $(x, y) = (0.0, 0.0)$ and $(x, y) = (100.0, 0.0)$ respectively. Node (1) is clamped, this is fixed in X, Y-directions and rotation. For dynamic analysis, the nodal force is applied at node (8) and is defined as $\sigma(t) = 8.75 \cdot 10^5 t \, \sin(0.4\pi \, t)$. The responses are verified against OpenSees using linear elastic beam with equivalent cross section parameters. Figure 79 shows teh vertical displacement at node (2) and the moment and shear forces developed at the fixed boundary.

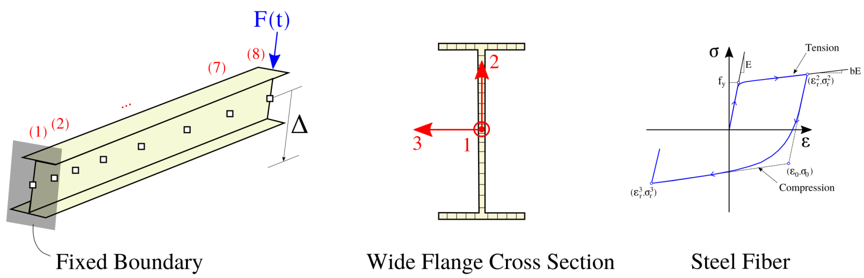


Figure 78: Varification for `Fib3DLineSection` section with `Steel1DFiber` fiber.

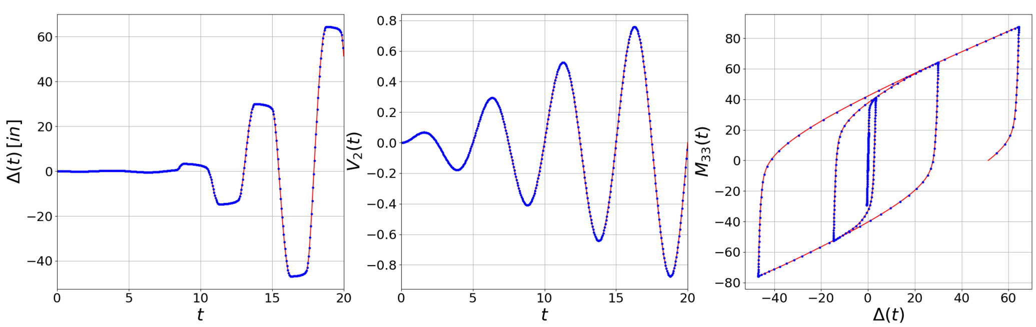


Figure 79: Nodal vertical responses at node (8) and shear force at node (8) : OpenSees (. . .), SeismoVLAB (——).

The root mean square error for the displacement is : `0.00465567`, while The maximum absolute difference for the displacement is : `0.0248980`.

DEBUG CASE : E01-ST_Lin_2DPointLoad_Elastic_Quad4

The problem showed in Figure 80 is a cantilever beam defined to test `lin2DQuad4` elements with material type `Elastic2DPlaneStress` and a point load. The material has a elasticity modulus $E = 100000\ Pa$, and a Poisson's ratio $\nu = 0.30$. Nodes (1) and node (11) have coordinate $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1) and (2) are fixed i.e, displacement in horizontal and vertical directions are fixed, while the others are free. The the element thickness is $t = 0.20\ m$, and the element height is $h = 0.20\ m$. The beam is subjected to a point load $q = 10\ [N]$ at Node (12). Responses are verified against analytical solution.

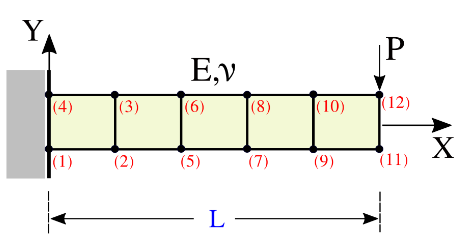


Figure 80: Varification for `lin2DQuad4` and Point Load in 2D.

The relative error for the vertical deformation at node (12) is : `0.308089`. The maximum relative error for the reaction forces is : `0.00000`.

DEBUG CASE : E02-ST_Lin_2DPointLoad_Elastic_Quad8

The problem showed in Figure 81 is a cantilever beam defined to test `lin2DQuad8` elements with material type `Elastic2DPlaneStress` and a point load. The material has a elasticity modulus $E = 100000\ Pa$, and a Poisson's ratio $\nu = 0.30$. Nodes (1) and node (11) have coordinate $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1) and (2) are fixed i.e, displacement in horizontal and vertical directions are fixed, while the others are free. The the element thickness is $t = 0.20\ m$, and the element height is $h = 0.20\ m$. The beam is subjected to a point load $q = 10\ [N]$ at Node (12). Responses are verified against analytical solution.

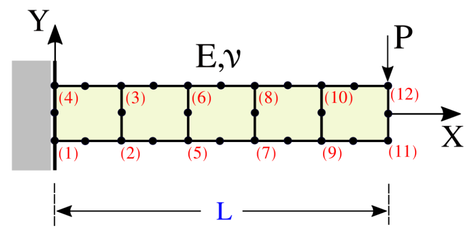


Figure 81: Varification for `lin2DQuad8` and Point Load in 2D.

The relative error for the vertical deformation at node (12) is : `0.006542`. The maximum relative error for the reaction forces is : `0.000000`.

DEBUG CASE : E03-ST_Lin_2DSurfaceLoad_Elastic_Quad4

The problem showed in Figure 82 is a cantilever beam defined to test `lin2DQuad4` elements with material type `Elastic2DPlaneStress` and a point load. The material has a elasticity modulus $E = 100000\ Pa$, and a Poisson's ratio $\nu = 0.30$. Nodes (1) and node (11) have coordinate $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1) and (2) are fixed i.e, displacement in horizontal and vertical directions are fixed, while the others are free. The the element thickness is $t = 0.20\ m$, and the element height is $h = 0.20\ m$. The beam is subjected to a distributed load $q = 10\ [N/m]$ at Node (12). Responses are verified against analytical solution.

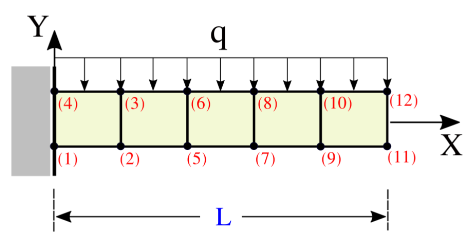


Figure 82: Varification for `lin2DQuad4` and Surface Load in 2D.

The relative error for the vertical deformation at node (12) is : `0.291526`. The maximum relative error for the reaction forces at node (1) is : `0.00000`.

DEBUG CASE : E04-ST_Lin_2DSurfaceLoad_Elastic_Quad8

The problem showed in Figure 83 is a cantilever beam defined to test `lin2DQuad8` elements with material type `Elastic2DPlaneStress` and a point load. The material has a elasticity modulus $E = 100000\ Pa$, and a Poisson's ratio $\nu = 0.30$. Nodes (1) and node (11) have coordinate $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1) and (2) are fixed i.e, displacement in horizontal and vertical directions are fixed, while the others are free. The the element thickness is $t = 0.20\ m$, and the element height is $h = 0.20\ m$. The beam is subjected to a distributed load $q = 10\ [N/m]$ at Node (12). Responses are verified against analytical solution.

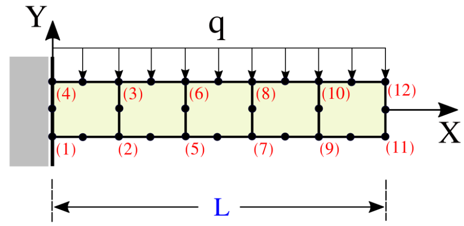


Figure 83: Varification for `lin2DQuad8` and Surface Load in 2D.

The relative error for the vertical deformation at node (12) is : `0.0159703`. The maximum relative error for the reaction forces at node (1) is : `0.00000`.

DEBUG CASE : E05-ST_Lin_2DRigidLink_Elastic_Frame2

The problem showed in Figure 84 is a portal frame defined to test `RigidLink` constraints. The material has a elasticity modulus $E = 100000\ Pa$, and a Poisson's ratio $\nu = 0.30$. Nodes (1) and node (5) have coordinate $(0.0, 0.0)$ and $(1.0, 1.0)$ respectively. Node (1) is clamped i.e, displacement and rotation are fixed, while nodes from (6) are free. Two rigid links are added between node (2)-(3) and (3)-(4) of length 0.001. The Bernoulli beam has a rectangular cross section with $h = b = 0.10\ m$. The beam is subjected to a vertical point load $P = 1\ [N]$. Responses are verified against analytical solution.

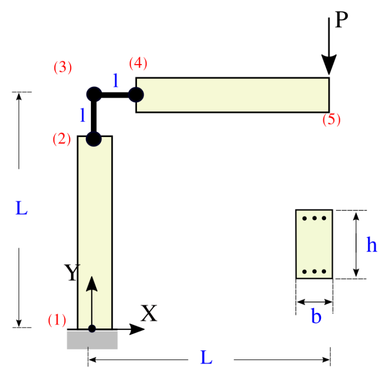


Figure 84: Varification for `RigidLink` constraint in 2D.

The relative error for the vertical deformation at node (5) is : `0.000874875`. The maximum relative error for the reaction forces at node (1) is : `0.00000`.

DEBUG CASE : F01-ST_Lin_2DPointLoad_ElasticPStrain_Quad4

The Problem is shown in Figure 85 and is defined to test `lin2DQuad4` element with material type provided in `Elastic2DPlaneStrain`. The material has $E = 208\ MPa$, $\nu = 0.3$, and $\rho = 2000\ kg/m^3$. Node (1) has coordinates $(0.0, 0.0)$ and is fixed in X and Y directions. Node (2) has coordinates $(2.0, 0.0)$ and is fixed in Y direction. Two nodal forces are placed at node (4) with $P_1 = 1\ MN$ and $P_2 = 2\ MN$. The responses are verified against OpenSees.

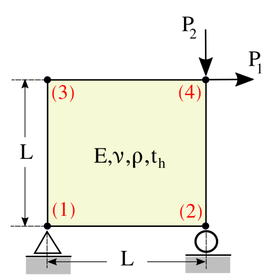


Figure 85: Verification for `lin2DQuad4` with `Elastic2DPlaneStrain` material.

The maximum absolute relative error for the displacements is : `0.00000`, while the maximum relative error for the strains and stresses are : `1.72395e-06`, `3.97755e-06` respectively.

DEBUG CASE : F02-DY_Lin_2DPointLoad_ElasticPStrain_Quad4

Problem setting is shown in Figure 85 and is defined to test `lin2DQuad4` element with material type provided in `Elastic2DPlaneStrain`. The material has $E = 208\ MPa$, $\nu = 0.3$, and $\rho = 2000\ kg/m^3$. Node (1) has coordinates $(0.0, 0.0)$ and is fixed in X and Y directions. Node (2) has coordinates $(2.0, 0.0)$ and is fixed in Y direction. Two nodal forces are prescribed at node (4) with $P_1 = 0.01 \cdot f(t)$ and $P_2 = 0.02 \cdot f(t)$ with $f(t) = 107.5 \cdot t\ \sin(2\pi\, t)$. The responses are verified against OpenSees. Figure 87 shows nodal displacements in X, Y directions at node (4).

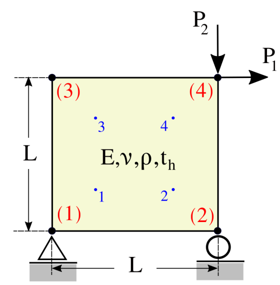


Figure 86: Varification for `lin2DQuad4` with `Elastic2DPlaneStrain` material.

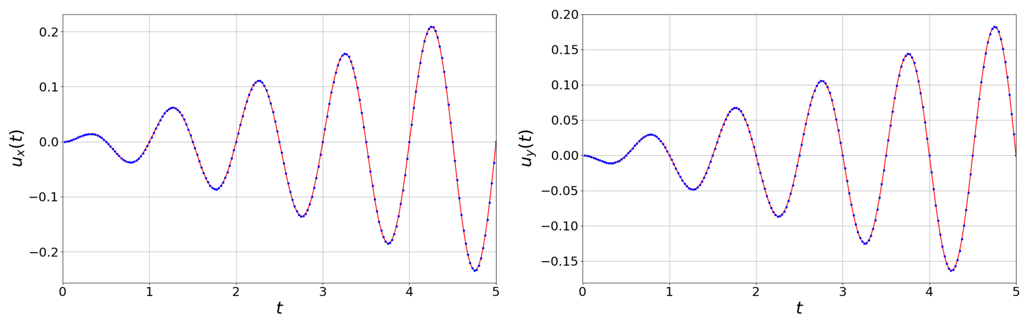


Figure 87: Nodal responses at node (4): OpenSEES (...), SeismoVLAB (——).

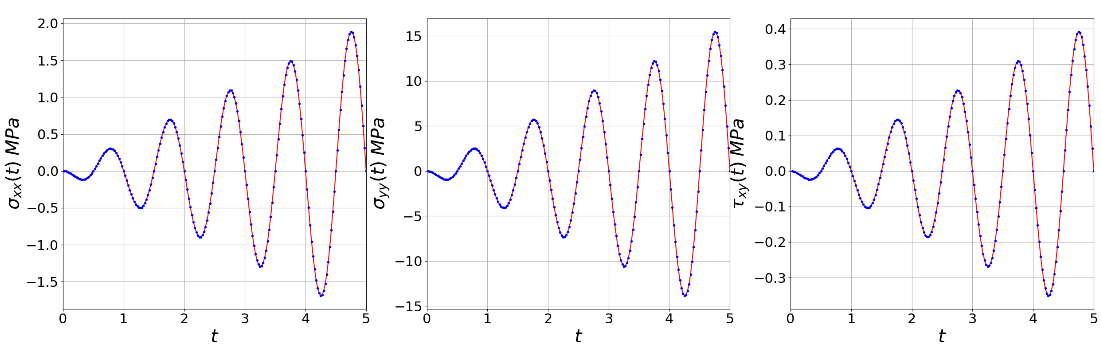


Figure 88: Material responses at integration point 4: OpenSEES (...), SeismoVLAB (——).

The root mean square error for displacements at node (4) is : (`1.84690e-07`, `1.47396e-07`), while the maximum relative error for the velocity and acceleration are : (`1.01859e-06`,`6.64316e-07`), and (`2.48365e-06`,`2.56551e-06`) respectively.

DEBUG CASE : F03-DY_Lin_2DPointLoad_J2PStrain_Quad4

Problem setting is shown in Figure 89 and is defined to test `lin2DQuad4` element with material type provided in `PlasticPlaneStrainJ2`. The material has $E = 208\ MPa$, $\nu = 0.3$, and $\rho = 2000\ kg/m^3$. Node (1) has coordinates $(0.0, 0.0)$ and is fixed in X and Y directions. Node (2) has coordinates $(2.0, 0.0)$ and is fixed in Y direction. Two nodal forces are prescribed at node (3) with $P_1 = 80 \cdot 10^6\, f(t)$ with $f(t)$ a ricker pulse. The responses are verified against OpenSees. Figure 87 shows nodal displacements in X and Y directions at node (4).

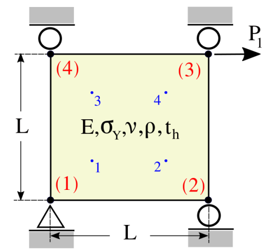


Figure 89: Varification for `lin2DQuad4` with `PlasticPlaneStrainJ2` material.

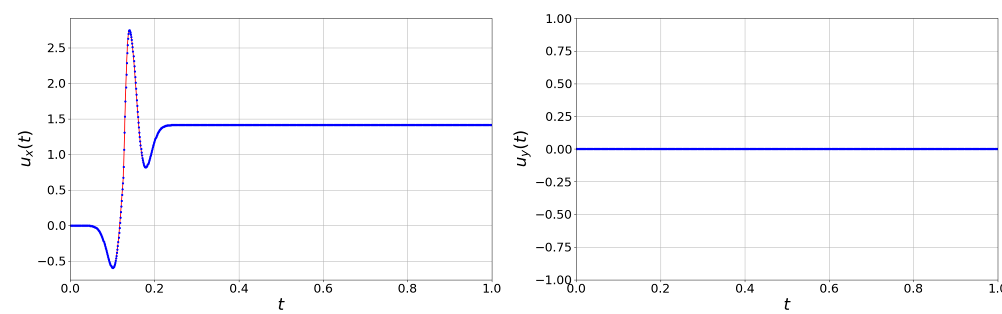


Figure 90: Nodal responses at node (4): OpenSEES (. . .), SeismoVLAB (——).

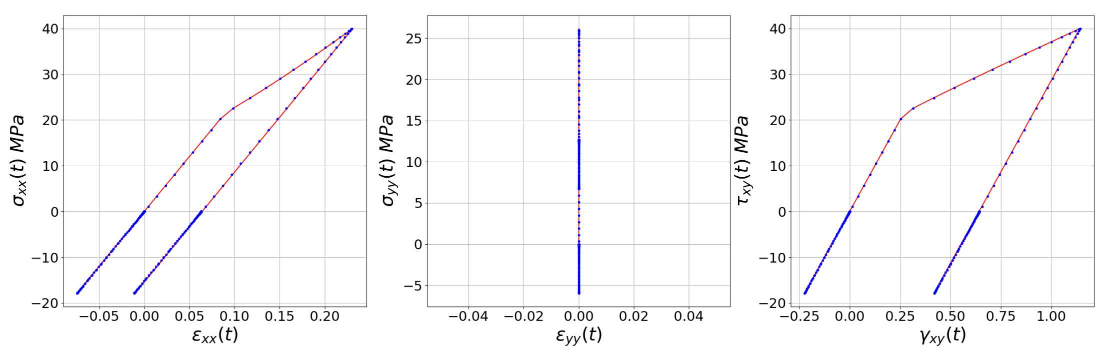


Figure 91: Material responses at integration point 4: OpenSEES (. . .), SeismoVLAB (——).

The root mean square for the relative displacements error at node (4) is : (`0.000891073, 0.00000`), while the root mean square for the relative error for the velocity and acceleration are : (`0.0204070,0.00000`), and (`0.0653151,0.00000`) respectively.

DEBUG CASE : F04-DY_Lin_2DPointLoad_BAPStrain_Quad4

Problem setting is shown in Figure 92 and is defined to test `lin2DQuad4` element with material type provided in `PlasticPlaneStrainBA`. The material has $E = 208\ MPa$, $\nu = 0.3$, and $\rho = 2000\ kg/m^3$. Node (1) has coordinates $(0.0, 0.0)$ and is fixed in X and Y directions. Node (2) has coordinates $(2.0, 0.0)$ and is fixed in X direction. Two nodal forces are prescribed at nodes (3) and (4) as $P_1 = 4 \cdot 10^4\, f(t)$ with $f(t)$ a ricker pulse. The responses are verified against OpenSees. Figure 87 shows nodal displacements in X and Y directions at node (4).

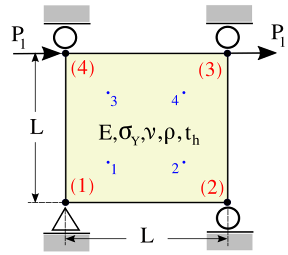


Figure 92: Varification for `lin2DQuad4` with `PlasticPlaneStrainJ2` material.

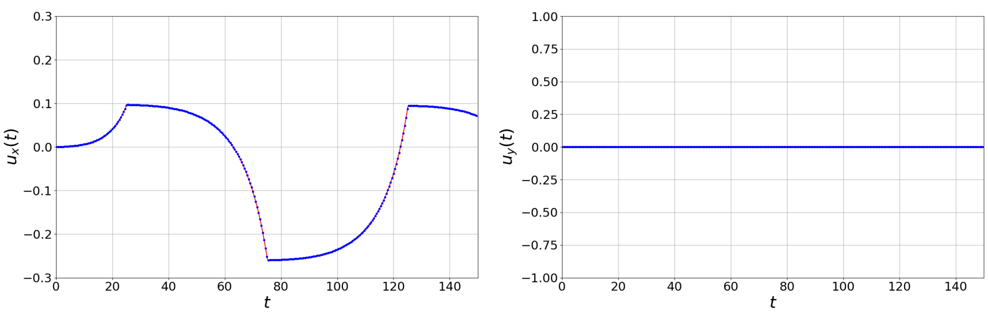


Figure 93: Nodal responses at node (4): OpenSEES (. . .), SeismoVLAB (——).

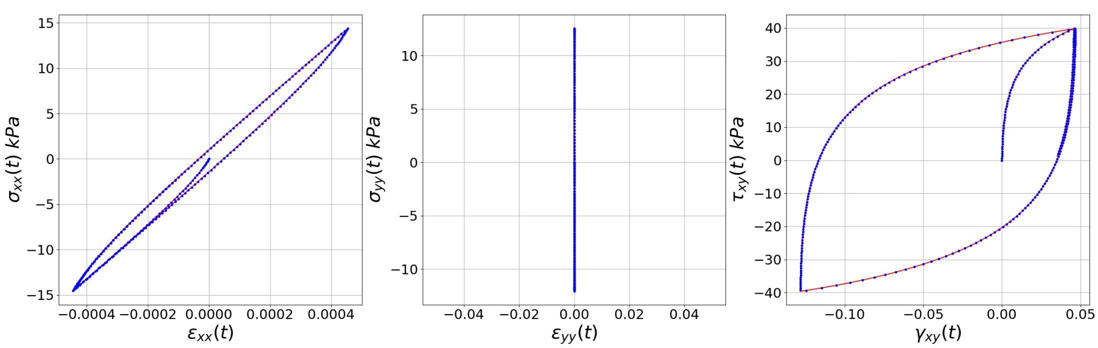


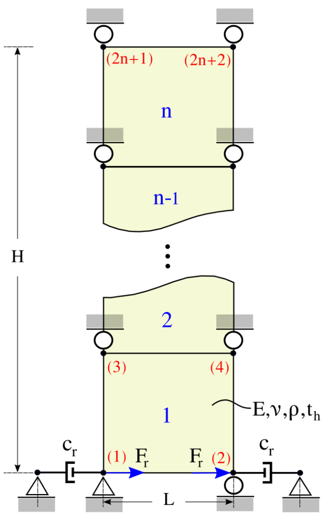
Figure 94: Material responses at integration point 4: OpenSEES (. . .), SeismoVLAB (——).

The root mean square error for the displacements at node (4) is : (`0.000146157`, `0.00000`), while the maximum relative error for the velocity and acceleration are : (`2.18573e-05`,`0.00000`), and (`0.00106666`,`0.00000`) respectively.

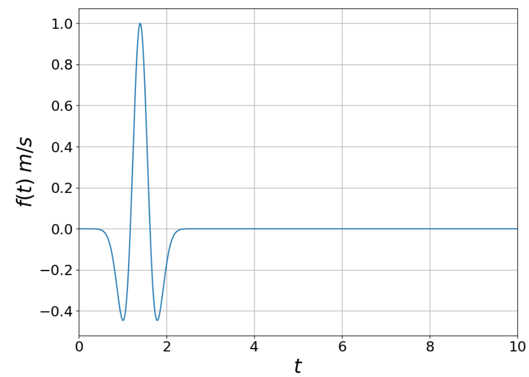
DEBUG CASE : F06-DY_Lin_2DSoilColumn_ElasticPStrain_Quad4

1D site response analysis in a truncated half-space can be modeled numerically using the setting shown in Figure 95a. We consider a soil column with height $H = 100\,m$ and width $B = 1\,m$. `linQuad2D4` element with `Elastic2DPlaneStrain` material is used for discretization. In total, 100 elements are used, i.e., $n = 100$. $E = 13\ MPa$, $\nu = 0.3$, and $\rho = 2000\ kg/m^3$. The half-space is truncated by using Lysmer dashpots and is modeled using `ZeroLength1D` element with `Viscous1DLinear` material. $c_r = \rho_r V_r B/2$ where $\rho_r = \rho$ and $V_r = 50\ m/s$. Incoming waves are modeled by nodal forces $f_r(t) = \rho_r V_r B/2 \cdot f(t)$ applied at nodes (1) and (2). The function $f(t)$

is the velocity of the incident motion at depth H and is shown in Figure 95b. Rayleigh damping is used to model small strain damping with mass and stiffness proportional coefficients of 0.12442 and 0.00079, respectively. The same problem is modeled in OpenSees for verification. Accelerations at nodes (1) and (201) are shown in Figure 96.



(a)



(b)

Figure 95: Varification for 1D site response analysis in an elastic half-space.

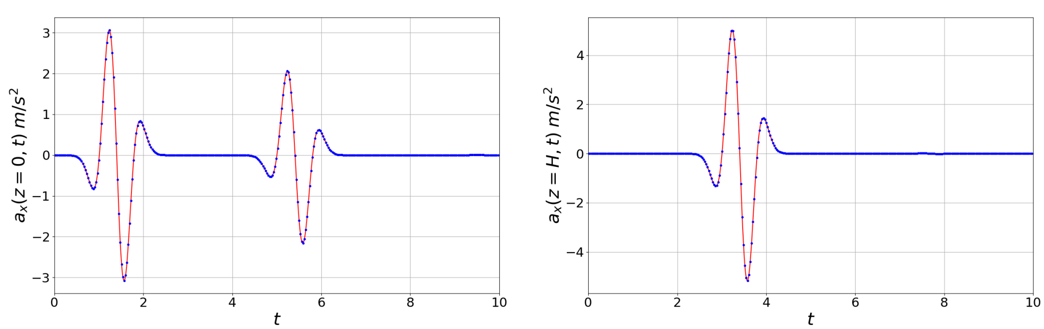
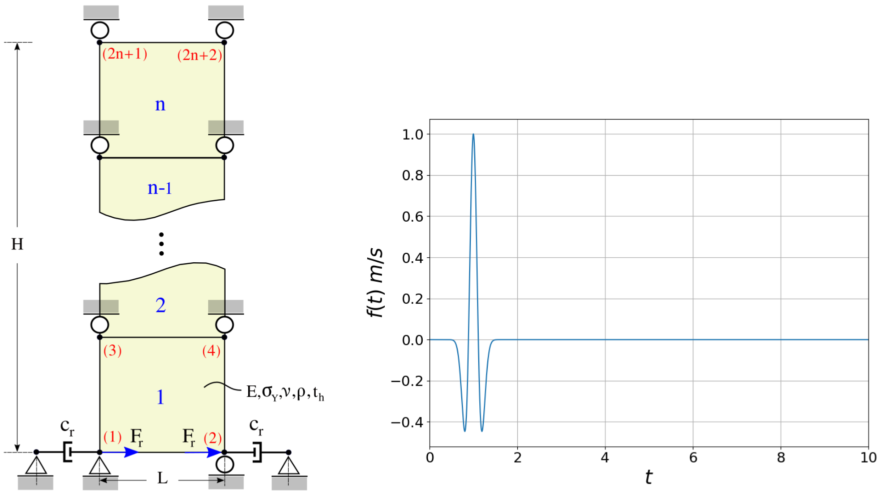


Figure 96: Accelerations at $z = H$ and $z = 0$: Opensees (...), SeismoVLAB (——).

The root mean square error for the displacements at node (201) is : (5.37488e-08, 0.00000), while the maximum relative error for the velocity and acceleration are : (1.01924e-07,0.00000), and (9.25660e-07,0.00000) respectively.

DEBUG CASE : F07-DY_Lin_2DSoilColumn_J2PStrain_Quad4

1D site response analysis in a truncated half-space can be modeled numerically using the setting shown in Figure 97a. We consider a soil column with height $H = 100\,m$ and width $B = 1\,m$. `linQuad2D4` element with `PlasticPlaneStrainJ2` material is used for discretization. In total, 100 elements are used, i.e., $n = 100$. $E = 13\ MPa$, $\nu = 0.3$, and $\rho = 2000\ kg/m^3$. The half-space is truncated by using Lysmer dashpots and is modeled using `ZeroLength1D` element with `Viscous1DLinear` material. $c_r = \rho_r V_r B/2$ where $\rho_r = \rho$ and $V_r = 50$ m/s. Incoming waves are modeled by nodal forces $f_r(t) = \rho_r V_r B/2 \cdot f(t)$ applied at nodes (1) and (2). The function $f(t)$ is the velocity of the incident motion at depth H and is shown in Figure 97b. Rayleigh damping is used to model small strain damping with mass and stiffness proportional coefficients of 0.12442 and 0.00079, respectively. The same problem is modeled in OpenSees for verification. Accelerations at nodes (1) and (201) are shown in Figure 98.



(a) (b)

Figure 97: Varification for 1D site response analysis in an elastic half-space.

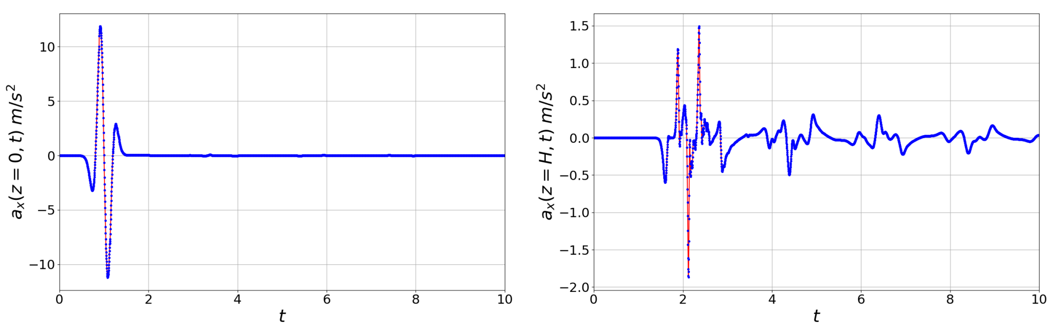


Figure 98: Accelerations at $z = H$ and $z = 0$: Opensees (. . .), SeismoVLAB (——).

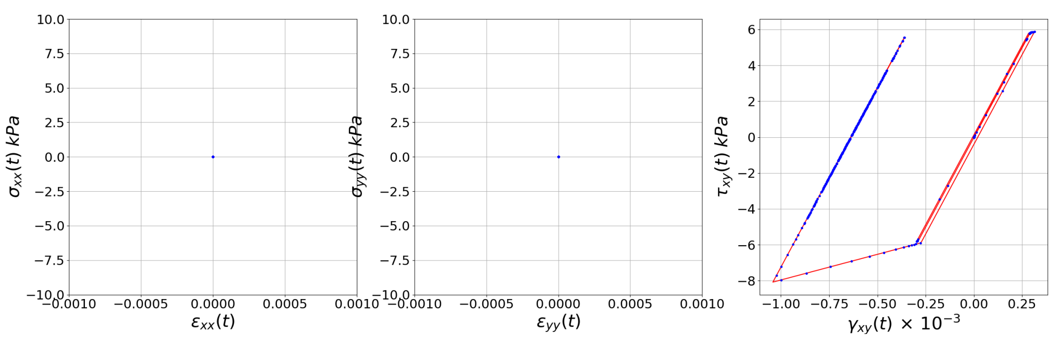


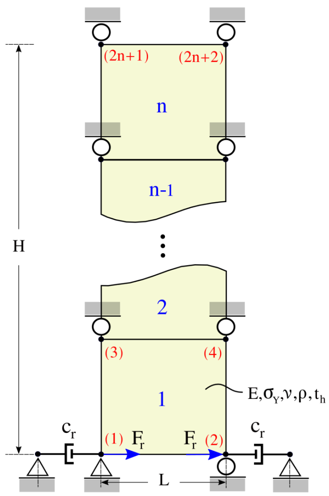
Figure 99: Material responses at element 50 integration point 4: OpenSEES (. . .), SeismoVLAB (——).

The root mean square error for the displacements at node (201) is : (`1.00530e-06`, `0.00000`), while the maximum relative error for the velocity and acceleration are : (`1.32156e-06`,`0.00000`), and (`2.42378e-05`,`0.00000`) respectively.

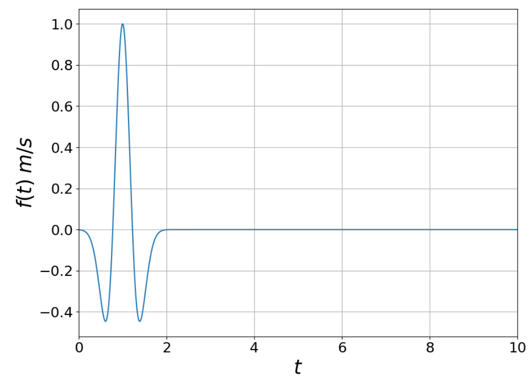
DEBUG CASE : F08-DY_Lin_2DSoilColumn_BAPStrain_Quad4

1D site response analysis in a truncated half-space can be modeled numerically using the setting shown in Figure 100a. We consider a soil column with height $H = 100\,m$ and width $B = 1\,m$. `linQuad2D4` element with

`PlasticPlaneStrainBA` material is used for discretization. In total, 100 elements are used, i.e., $n = 100$. $E = 13\ MPa$, $\nu = 0.3$, and $\rho = 2000\ kg/m^3$. The half-space is truncated by using Lysmer dashpots and is modeled using `ZeroLength1D` element with `Viscous1DLinear` material. $c_r = \rho_r V_r B/2$ where $\rho_r = \rho$ and $V_r = 50$ m/s. Incoming waves are modeled by nodal forces $f_r(t) = \rho_r V_r B/2 \cdot f(t)$ applied at nodes (1) and (2). The function $f(t)$ is the velocity of the incident motion at depth H and is shown in Figure 100b. Rayleigh damping is used to model small strain damping with mass and stiffness proportional coefficients of 0.12442 and 0.00079, respectively. The same problem is modeled in OpenSees for verification. Accelerations at nodes (1) and (201) are shown in Figure 101.



(a)



(b)

Figure 100: Varification for 1D site response analysis in an elastic half-space.

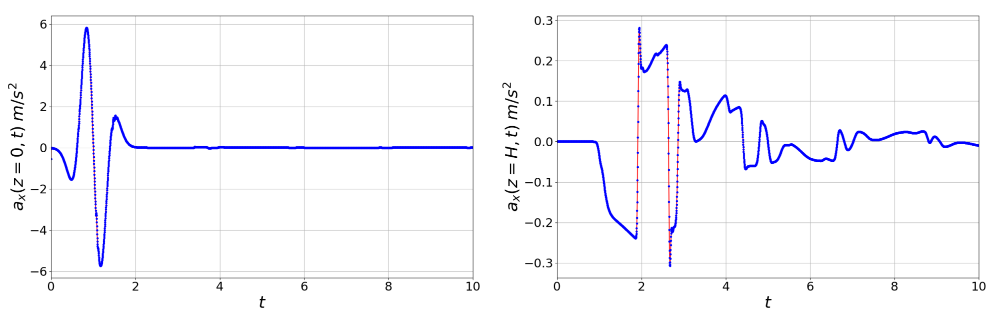


Figure 101: Accelerations at $z = H$ and $z = 0$: Opensees (...), SeismoVLAB (——).

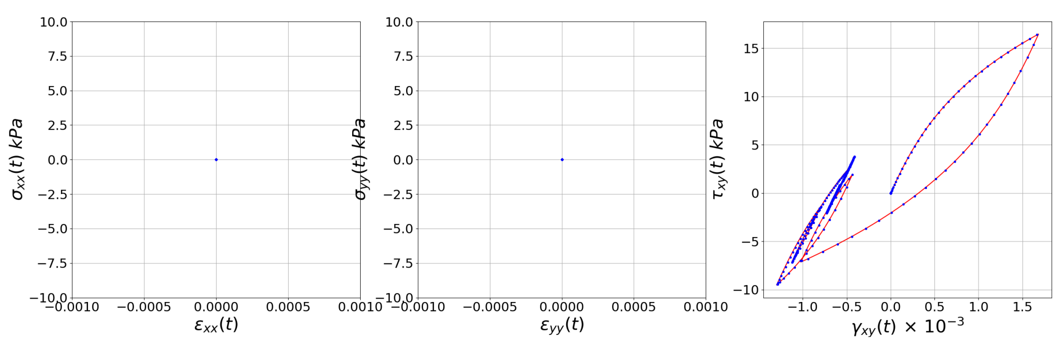


Figure 102: Material responses at element 50 integration point 4: OpenSEES (...), SeismoVLAB (——).

The root mean square error for the displacements at node (201) is : (0.000232257, 0.00000), while the maximum relative error for the velocity and acceleration are : (0.000237063,0.00000), and (0.00273708,0.00000) respectively.

DEBUG CASE : F11-DY_Lin_2DPMLSoilColumn_ElasticPStrain_Quad4

The Problem is shown in Figure 103 and is defined to test `PML2DQuad4` element with material type provided in `Elastic2DPlaneStrain`. The material has $E = 50\ MPa$, $\nu = 0.25$, and $\rho = 2000\ kg/m^3$. The rod is 100 m long and is fixed on the left hand side and isfree to move in the axial direction. Two nodal forces are placed on the right end with $P_1 = 0.1\ MN$. The velocity responses are evaluated at the right border (blue) and PML/Quad interface (red) and they should show no reflections after $t \geq 2\ [s]$.

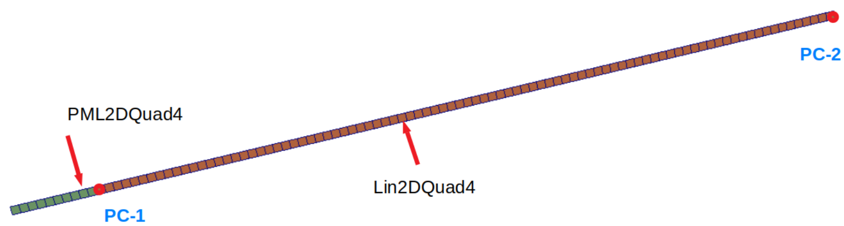


Figure 103: Verification for `PML2DQuad4` with `Elastic2DPlaneStrain` material.

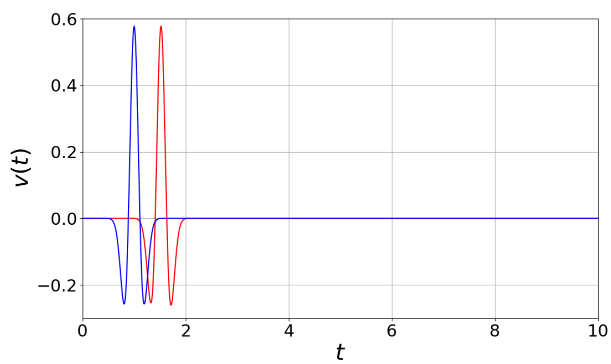


Figure 104: Nodal velocity responses in SeismoVLAB (blue PC_2) (red PC_1).

DEBUG CASE : F14-ST_Lin_2DConstrainedSSI_Elastic_Quad4_Frame2

The problem showed in Figure 105 is a portal frame supported on a linear elastic soil in plain strain defined to test `lin2DFrame2` and `lin2DQuad4` elements with material type `Elastic1DLinear` and `Elastic2DPlaneStrain` respectively, and kinemtic constraints between solid and strcutural elements. The soil material has a elasticity moduli $E = 50.0\,MPa$, and a Poisson's ratio $\nu = 0.25$ and beam and column material has a elasticity moduli $E = 25.0\,GPa$, and a Poisson's ratio $\nu = 0.25$. Nodes (3322), (3348) and (3374) have coordinate $(5.0, 10.0)$, $(10.0, 10.0)$, and $(15.0, 10.0)$ respectively. All nodes at the boundary represented by a thick solid-black line are fixed in X and Y directions. The Bernoulli beam is employed to model columns and beams with rectangular cross section $h = 0.9\,m$ and $b = 0.6\ m$. A horizontal load is placed at node (3347) with magnitude $P = 100\ kN$, and kinematic constraints are enforced on the horizontal and vertical direction for all nodes in the interface column and soil. Responses are verified against numerical (OpenSees) solution.

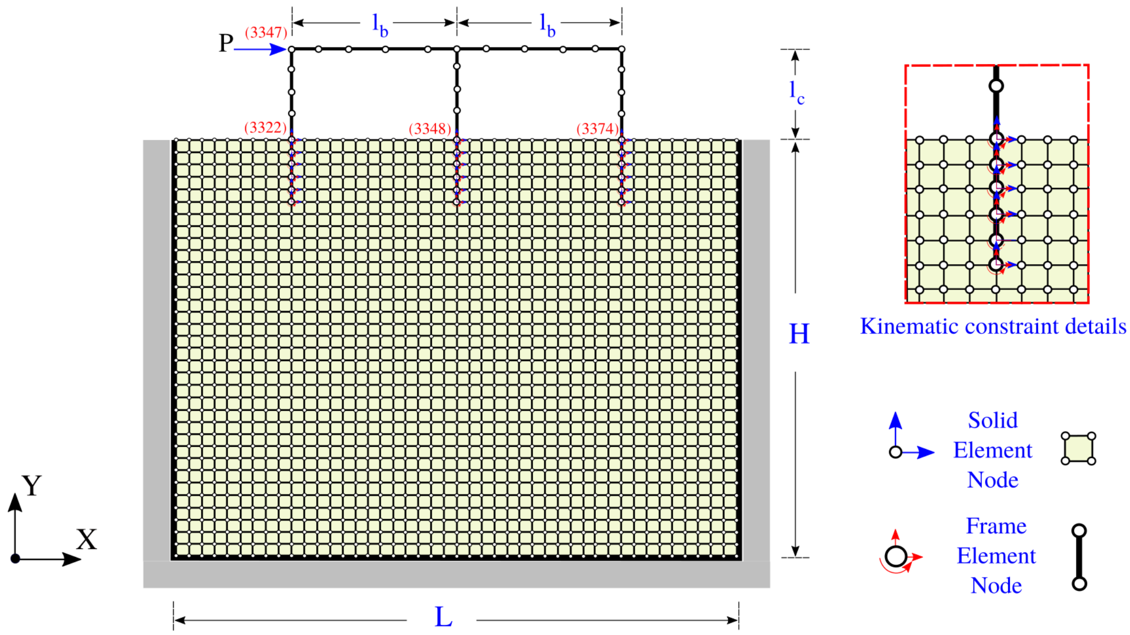


Figure 105: Varification for `lin2DFrame2` with `Elastic1DLinear` material and kinematic constraints.

The relative error for the horizontal deformation at node (3347) is : `1.19372e-07`. The maximum relative error for the reaction forces at node (3322), (3348), and (3374) are : `2.70805e-06`.

DEBUG CASE : H01-ST_Lin_3DinPlanePointLoad_ElasticPStress_Shell4

The problem showed in Figure 106 is a vertical shell (wall) element defined to test `lin3DShell4` elements with material type `Elastic2DPlaneStress`. The wall element has material with elasticity moduli $E = 25.0\,GPa$, and a Poisson's ratio $\nu = 0.25$. Nodes (1), (2), (3), and (4) have coordinate $(5.0, 0.0, 0.0)$, $(0.0, 0.0, 0.0)$, and $(0.0, 0.0, 5.0)$, and $(5.0, 0.0, 5.0)$ respectively. Nodes (1) and (2) are clamped, i.e., displacements and rotation along/about X, Y and Z are restrained. The shell has a thickness of $t_h = 0.1\ m$. A horizontal in-plane load is placed at node (4) with magnitude $P = 10\ kN$ and direction $\hat{n} = (-1, 0, 0)$. Responses are verified against numerical (ETABS) solution.

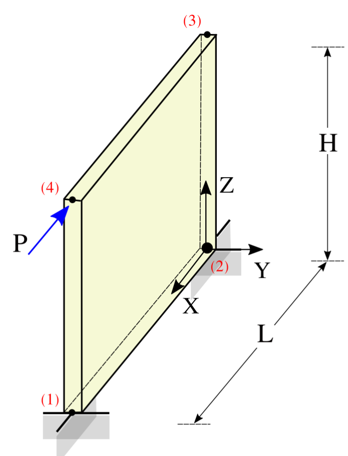


Figure 106: Varification for `lin3DShell4` with `Elastic2DPlaneStress` material and kinematic constraints.

The relative error for the horizontal deformation at node (3) and (4) is : `0.0937416`. The maximum relative error for the reaction forces at node (1) and (2) is : `0.418131`.

DEBUG CASE : H02-ST_Lin_3DoutPlanePointLoad_ElasticPStress_Shell4

The problem showed in Figure 107 is a vertical shell (wall) element defined to test `lin3DShell4` elements with material type `Elastic2DPlaneStress`. The wall element has material with elasticity moduli $E = 25.0\,GPa$, and a Poisson's ratio $\nu = 0.25$. Nodes (1), (2), (3), and (4) have coordinate $(5.0, 0.0, 0.0)$, $(0.0, 0.0, 0.0)$, and $(0.0, 0.0, 5.0)$, and $(5.0, 0.0, 5.0)$ respectively. Nodes (1) and (2) are clamped, i.e., displacements and rotation along/about X, Y and Z are restrained. The shell has a thickness of $t_h = 0.1\ m$. A horizontal out-plane load is placed at node (4) with magnitude $P = 10\ kN$ and direction $\hat{n} = (0, 1, 0)$. Responses are verified against numerical (ETABS) solution.

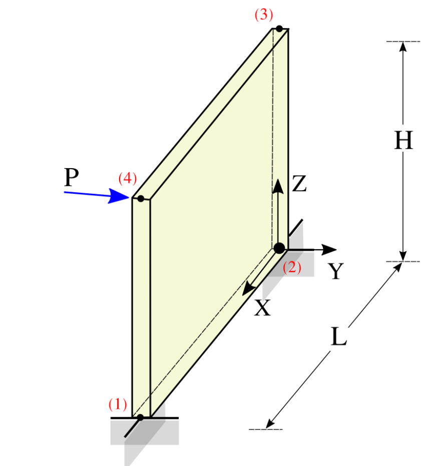


Figure 107: Varification for `lin3DShell4` with `Elastic2DPlaneStress` material and kinematic constraints.

The relative error for the horizontal deformation at node (3) and (4) is : `0.0109294`. The maximum relative error for the reaction forces at node (1) and (2) is : `0.00847829`.

DEBUG CASE : H03-ST_Lin_3DSlabPointLoad_ElasticPStress_Shell4

The problem showed in Figure 109 is a horizontal shell (slab) elements defined to test `lin3DShell4` elements with material type `Elastic2DPlaneStress`. The slab elements have material with elasticity moduli $E = 25.0\,GPa$, and a Poisson's ratio $\nu = 0.25$. Nodes (1), (2), (3), and (4) have coordinate $(0.0, 0.0, 0.0)$, $(6.0, 0.0, 0.0)$, and $(6.0, 6.0, 0.0)$, and $(0.0, 6.0, 0.0)$ respectively. Nodes at the boundaries are clamped, i.e., displacements and rotation along/about X, Y and Z are restrained. The shell has a thickness of $t_h = 0.1\ m$. A vertical out-plane load is placed at node (25) with magnitude $P = 10\ kN$ and direction $\hat{n} = (0, 0, -1)$. Responses are verified against numerical (ETABS) solution.

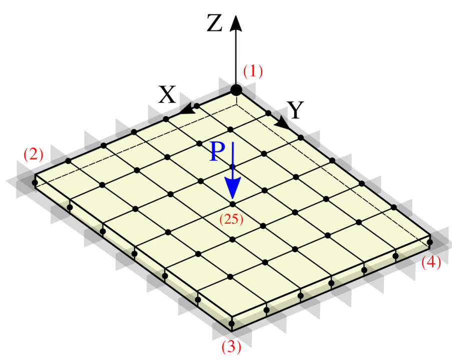


Figure 108: Varification for `lin3DShell4` with `Elastic2DPlaneStress` material and kinematic constraints.

The relative error for the horizontal deformation at node (25), (26) and (32) is : `0.00123223`.

DEBUG CASE : H04-ST_Lin_3DSlabBodyLoad_ElasticPStress_Shell4

The problem showed in Figure 109 is a horizontal shell (slab) elements defined to test `lin3DShell4` elements with material type `Elastic2DPlaneStress`. The slab elements have material with elasticity moduli $E = 25.0\,GPa$, and a Poisson's ratio $\nu = 0.25$. Nodes (1), (2), (3), and (4) have coordinate $(0.0, 0.0, 0.0)$, $(6.0, 0.0, 0.0)$, and $(6.0, 6.0, 0.0)$, and $(0.0, 6.0, 0.0)$ respectively. Nodes at the boundaries are clamped, i.e., displacements and rotation along/about X, Y and Z are restrained. The shell has a thickness of $t_h = 0.1\ m$. A vertical out-plane load is placed at node (25) with magnitude $P = 10\ kN$ and direction $\hat{n} = (0, 0, -1)$. Responses are verified against numerical (ETABS) solution.

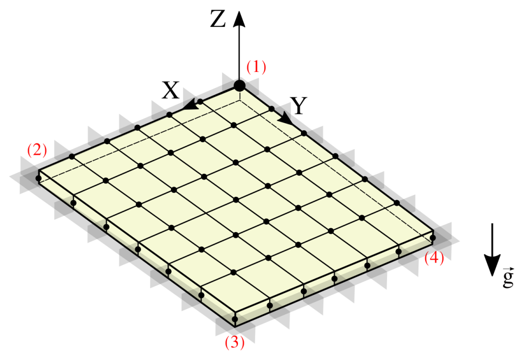


Figure 109: Varification for `lin3DShell4` with `Elastic2DPlaneStress` material and kinematic constraints.

The relative error for the horizontal deformation at node (25), (26) and (32) is : `0.00148061`.

DEBUG CASE : H05-ST_Lin_3DBuildingDiaphragm_ElasticPStress_Frame2_Shell4

The problem showed in Figure 110 is a one-story reinforced-concrete building defined to test `lin3DShell4` and `lin3DFrame2` elements with material type `Elastic2DPlaneStress` and `Elastic1DLinear` respectively. The shell element has material with elasticity moduli $E = 25.0\,GPa$, and a Poisson's ratio $\nu = 0.25$ as well as the columns. Nodes (1), (2), (3), and (4) have coordinate $(0.0, 0.0, 0.0)$, $(5.0, 0.0, 0.0)$, and $(5.0, 5.0, 0.0)$, and $(0.0, 5.0, 0.0)$ respectively, and they are clamped, i.e., displacements and rotation along/about X, Y and Z are restrained. Nodes (5), (6), (7), and (8) are constrained by a rigid diaphragm acting on the +Z axis. The shell element has a thickness of $t_h = 0.1\,m$ and the frame elements have a cross section of $h = 0.75\,m$ and $b = 0.25\,m$. A horizontal in-plane load is placed at node (8) with magnitude $P = 10\ kN$ and direction $\hat{n} = (1, 0, 0)$. Responses are verified against numerical (ETABS) solution.

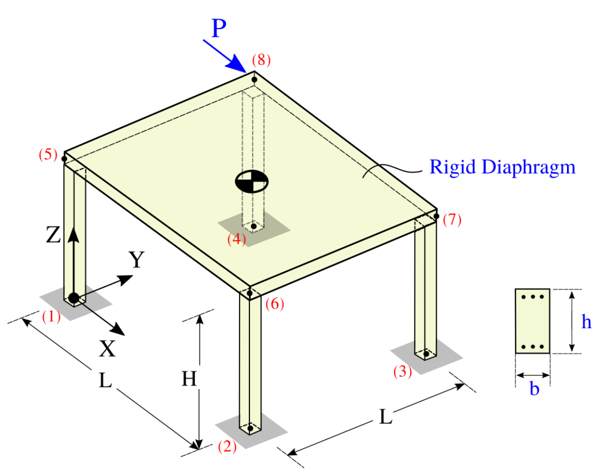


Figure 110: Varification for `lin3DShell4` with `Elastic2DPlaneStress` material and kinematic constraints.

The relative error for the displacement/rotations at node (5), (6), (7) and (8) is : `0.994092`. The maximum relative error for the reaction forces at node (1), (2), (3) and (4) is : `1.64325`.

DEBUG CASE : H06-ST_Lin_3DSlabSurfaceLoad_ElasticPStress_Shell4

The problem showed in Figure 111 is a horizontal shell (slab) elements defined to test `lin3DShell4` elements with material type `Elastic2DPlaneStress`. The slab elements have material with elasticity moduli $E = 25.0\ GPa$, and a Poisson's ratio $\nu = 0.25$. Nodes (1), (2), (3), and (4) have coordinate $(0.0, 0.0, 0.0)$, $(6.0, 0.0, 0.0)$, and $(6.0, 6.0, 0.0)$, and $(0.0, 6.0, 0.0)$ respectively. Nodes at the boundaries are clamped, i.e., displacements and rotation along/about X, Y and Z are restrained. The shell has a thickness of $t_h = 0.1\ m$. A vertical out-plane surface load is added with magnitude $q = 2450\ N/m$ and direction $\hat{n} = (0, 0, -1)$. Responses are verified against numerical (ETABS) solution.

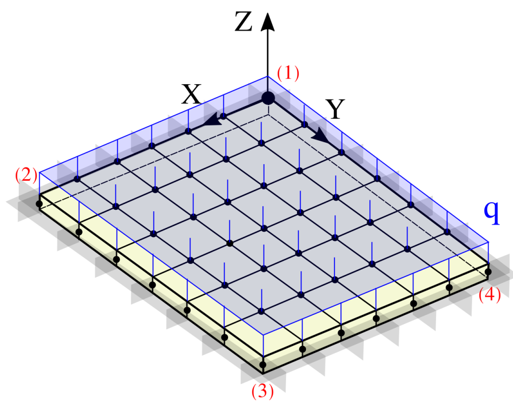


Figure 111: Verification for `lin3DShell4` with `Elastic2DPlaneStress` material and Surface Load.

The relative error for the horizontal deformation at node (25), (26) and (32) is : `0.00148061`.

DEBUG CASE : H07-ST_Lin_3DCantileverSurfaceLoad_Shell4

The problem showed in Figure 112 is a vertical shell (wall) elements defined to test `lin3DShell4` elements with material type `Elastic2DPlaneStress`. The slab elements have material with elasticity moduli $E = 100000.0\ Pa$, and a Poisson's ratio $\nu = 0.30$. Nodes (1), (2), (11), and (12) have coordinate $(0.0, 0.0, 0.0)$, $(0.2, 0.0, 0.0)$, and $(0.2, 0.0, 1.0)$, and $(0.0, 0.0, 1.0)$ respectively. Nodes (1) and (2) at the boundaries are clamped, i.e., displacements and rotation along/about X, Y and Z are restrained. The shell has a thickness of $t_h = 0.2\ m$. A horizontal out-plane surface load is added with magnitude $q = 50\ N/m^2$ and direction $\hat{n} = (0, 1, 0)$. Responses are verified against analytical solution.

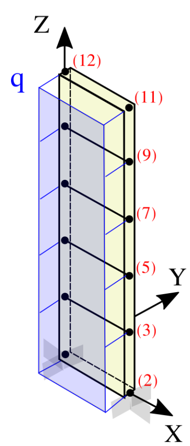


Figure 112: Verification for `lin3DShell4` with `Elastic2DPlaneStress` material and Surface Load.

The relative error for the horizontal deformation at node (11) and (12) is : `0.0102621`. The maximum relative error for the reaction forces at node (1) is : `0.00000`.

DEBUG CASE : H08-DY_Damped_3DPointLoad_Plate_Elastic_Shell4

Problem setting is shown in Figure 113 and is defined to test `lin3DShell4` element with material type `Elastic2DPlaneStress`. For this example, all shell members have a rectangular cross-section with thickness $t = 1$, and modulus of elasticity, $E = 35000$. The beam is 10 long and is discretized with 10 equal length shell elements and 22 nodes. A vertical dynamic load is placed at Node (11) and (22) of constant magnitude 5. Rayleigh damping is added with $a_0 = 0$, and $a_1 = 0.02$. The responses are verified against analytical solution. Figure 114 shows the force displacement curve at node (11).

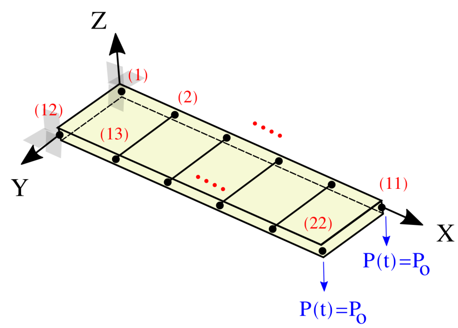


Figure 113: Varification for `lin3DShell4` with `Lin3DThinArea` Section.

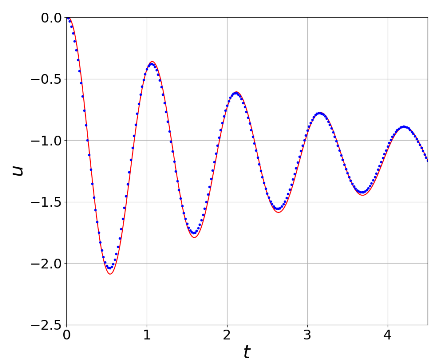


Figure 114: Force displacement curve at (11): Analytical (. . .), SeismoVLAB (——).

DEBUG CASE : I01-ST_Lin_3DPointLoad_Elastic_Hexa8

The problem showed in Figure 115 is a cantilever beam defined to test `lin3DHexa8` elements with material type `Elastic3DLinear` and a point load. The material has a elasticity modulus $E = 100000\ Pa$, and a Poisson's ratio $\nu = 0.30$. Nodes (1) and node (21) have coordinate $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1), (2), (3) and (4) are fixed i.e, displacement in horizontal and vertical directions are fixed, while the others are free. The the element thickness is $t = 0.20\ m$, and the element height is $h = 0.20\ m$. The beam is subjected to a point load $P = 5\ [N]$ at Node (23) and (24). Responses are verified against analytical solution.

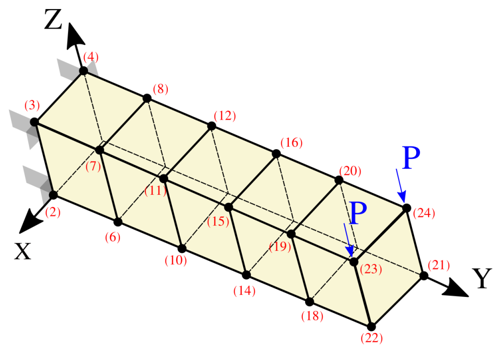


Figure 115: Varification for `lin3DHexa8` and Point Load in 3D.

The relative error for the vertical deformation at node (24) is : `0.342711`. The maximum relative error for the reaction forces is : `0.00000`.

DEBUG CASE : I02-ST_Lin_3DSurfaceLoad_Elastic_Hexa8

The problem showed in Figure 119 is a cantilever beam defined to test `lin3DHexa8` elements with material type `Elastic3DLinear` and a surface load. The material has a elasticity modulus $E = 100000\ Pa$, and a Poisson's ratio $\nu = 0.30$. Nodes (1) and node (21) have coordinate $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1), (2), (3) and (4) are fixed i.e, displacement in horizontal and vertical directions are fixed, while the others are free. The the element thickness is $t = 0.20\ m$, and the element height is $h = 0.20\ m$. The beam is subjected to a point load $q = 10\ [N]$ at Node (24). Responses are verified against analytical solution.

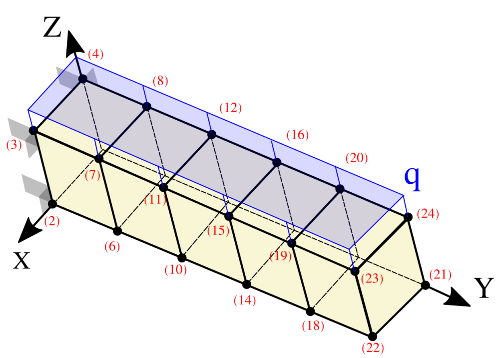


Figure 116: Varification for `lin2DQuad4` and Point Load in 2D.

The relative error for the vertical deformation at node (24) is : `0.332143`. The maximum relative error for the reaction forces is : `0.00000`.

DEBUG CASE : I03-ST_Lin_3DPointLoad_Elastic_Hexa20

The problem showed in Figure 117 is a cantilever beam defined to test `lin3DHexa20` elements with material type `Elastic3DLinear` and a point load. The material has a elasticity modulus $E = 100000\ Pa$, and a Poisson's ratio $\nu = 0.30$. Nodes (1) and node (6) have coordinate $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1), (7), (13), (19), (30), (46), (57), and (63) are fixed i.e, displacement in horizontal and vertical directions are fixed, while the others are free. The the element thickness is $t = 0.20\ m$, and the element height is $h = 0.20\ m$. The beam is subjected to a point load $P = 5\ [N]$ at Node (18) and (24). Responses are verified against analytical solution.

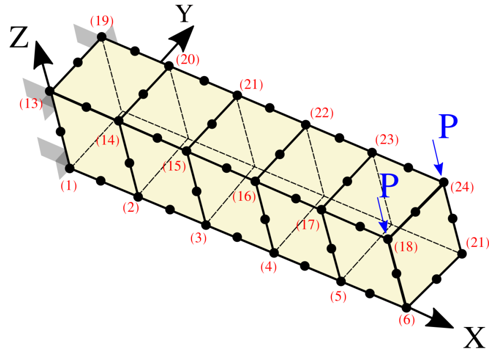


Figure 117: Varification for `lin3DHexa20` and Point Load in 3D.

The relative error for the vertical deformation at node (24) is : `0.00309956`. The maximum relative error for the reaction forces is : `4.00000e-09`.

DEBUG CASE : I04-ST_Lin_3DSurfaceLoad_Elastic_Hexa20

The problem showed in Figure 120 is a cantilever beam defined to test `lin3DHexa20` elements with material type `Elastic3DLinear` and a surface load. The material has a elasticity modulus $E = 100000\ Pa$, and a Poisson's ratio $\nu = 0.30$. Nodes (1) and node (6) have coordinate $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1), (7), (13), (19), (30), (46), (57), and (63) are fixed i.e, displacement in horizontal and vertical directions are fixed, while the others are free. The the element thickness is $t = 0.20\ m$, and the element height is $h = 0.20\ m$. The beam is subjected to a surface load $q = 50\ [N/m^2]$ at Node (18) and (24). Responses are verified against analytical solution.

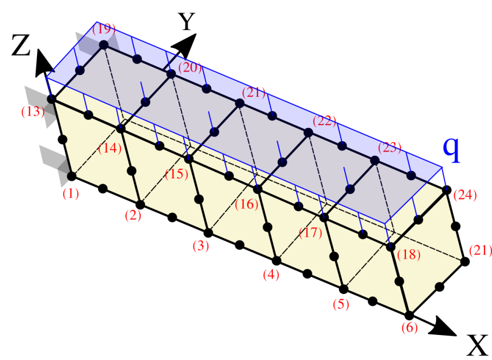


Figure 118: Varification for `lin3DHexa20` and Surface Load in 3D.

The relative error for the vertical deformation at node (24) is : `0.0113384`. The maximum relative error for the reaction forces is : `1.00000e-09`.

DEBUG CASE : I05-ST_Lin_3DBodyLoad_Elastic_Hexa8

The problem showed in Figure 119 is a cantilever beam defined to test `lin3DHexa8` elements with material type `Elastic3DLinear` and body load. The material has a elasticity modulus $E = 100000\ Pa$, a Poisson's ratio $\nu = 0.30$, and material density $ho = 25\ [kg/m^3]$. Nodes (1) and node (6) have coordinate $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1), (2), (3), and (4) are fixed i.e, displacement in horizontal and vertical directions are fixed, while the others are free. The the element thickness is $t = 0.20\ m$, and the element height is $h = 0.20\ m$. The beam is subjected to a gravity field of $g = 10\ [m/s^2]$. Responses are verified against analytical solution.

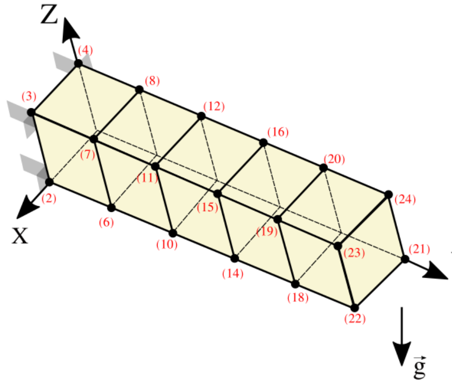


Figure 119: Varification for `lin3DHexa8` and Body Load in 3D.

The relative error for the vertical deformation at node (24) is : `0.332409`. The maximum relative error for the reaction forces is : `0.00000`.

DEBUG CASE : I06-ST_Lin_3DBodyLoad_Elastic_Hexa20

The problem showed in Figure 120 is a cantilever beam defined to test `lin3DHexa20` elements with material type `Elastic3DLinear` and body load. The material has a elasticity modulus $E = 100000\ Pa$, a Poisson's ratio $\nu = 0.30$, and material density $ho = 25\ [kg/m^3]$. Nodes (1) and node (6) have coordinate $(0.0, 0.0)$ and $(1.0, 0.0)$ respectively. Node (1), (7), (13), (19), (30), (46), (57), and (63) are fixed i.e, displacement in horizontal and vertical directions are fixed, while the others are free. The the element thickness is $t = 0.20\ m$, and the element height is $h = 0.20\ m$. The beam is subjected to a gravity field of $g = 10\ [m/s^2]$. Responses are verified against analytical solution.

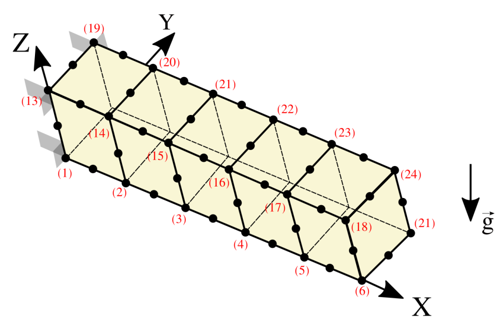


Figure 120: Varification for `lin3DHexa20` and Body Load in 3D.

The relative error for the vertical deformation at node (24) is : `0.0152606`. The maximum relative error for the reaction forces is : `2.00000e-09`.

DEBUG CASE : J02-DY_Lin_3DPointLoad_Elastic_Hexa8

Problem setting is shown in Figure 121 and is defined to test `lin3DHexa8` element with material type provided in `Elastic3DLinear`. The material has $E = 208\,MPa$, $\nu = 0.3$, and $\rho = 2000\ kg/m^3$. Node (1) has coordinates $(0.0, 0.0, 0.0)$ and is fixed in X, Y and Z directions. Node (2), (3), and (4) have coordinates $(L, 0.0, 0.0)$, $(L, L, 0.0)$ and $(0.0, L, 0.0)$ with $L = 2\,m$, and they are fixed in Z direction. For dynamic analysis, the nodal forces are defined as $F_1 = 0.01 \cdot f(t)$, $F_2 = 0.01 \cdot f(t)$, and $F_3 = 0.05 \cdot f(t)$ with $f(t) = 107.5 \cdot t\,\sin(2\pi\,t)$. The responses are verified against OpenSees. Figure 122 shows nodal displacements in X, Y, and Z directions at node (7).

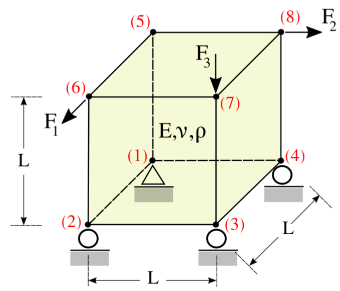


Figure 121: Verification for extttlin3DHexa8 with `Elastic3DLinear` material.

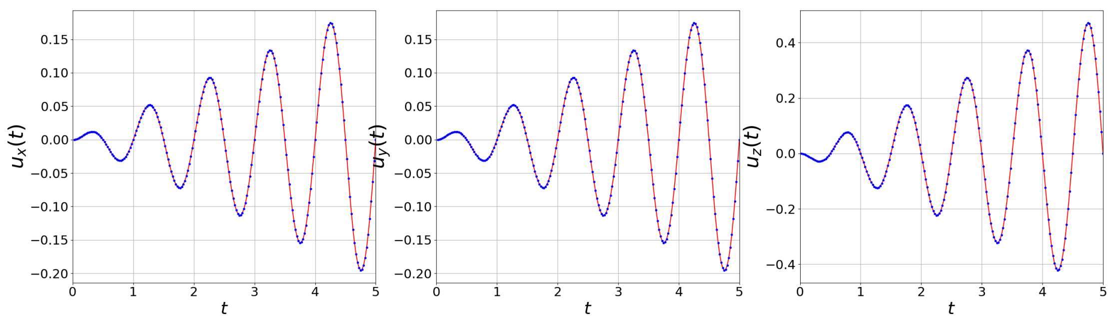


Figure 122: Nodal responses at node (4): OpenSEES (. . .), SeismoVLAB (——).

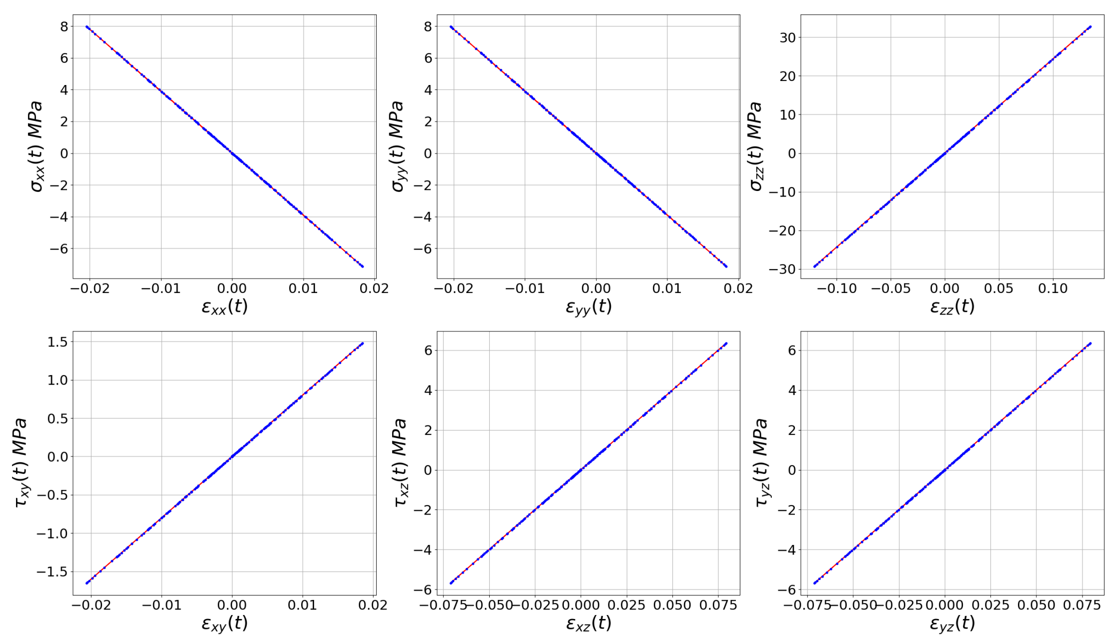
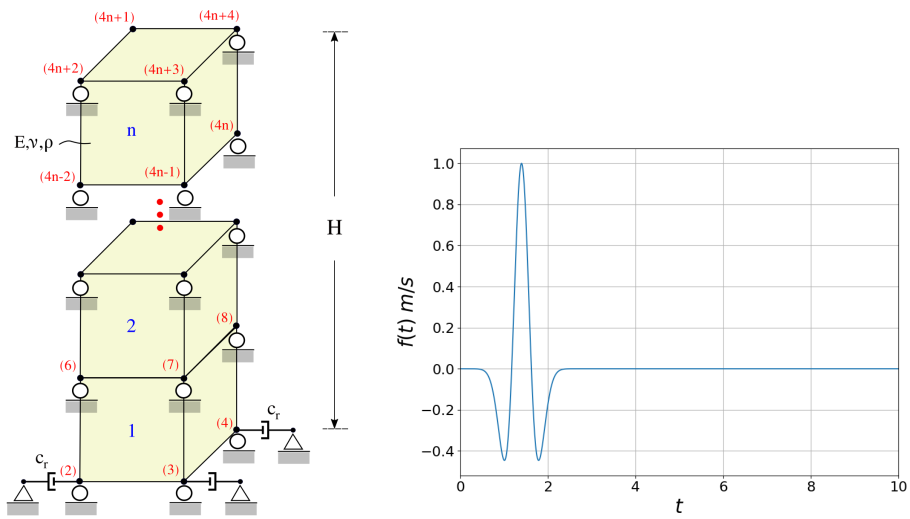


Figure 123: Material responses at integration point 4: OpenSEES (. . .), SeismoVLAB (——).

The root mean square error for the displacements at node (7) is : (`1.52790e-07`, `1.52790e-07`, `2.11009e-07`), while the maximum relative error for the velocity and acceleration are : (`9.33439e-07`,`9.33439e-07`,`2.00149e-06`), and (`2.63210e-06`,`2.63210e-06`,`1.46572e-05`) respectively.

DEBUG CASE : J05-DY_Lin_3DSoilColumn_Elastic_Hexa8

1D site response analysis in a truncated half-space can be modeled numerically using the setting shown in Figure 124a. We consider a soil column with height $H = 100\,m$ length $L = 1\,m$ and width $B = 1\,m$. `lin3DHexa8` element with `Elastic3DLinear` material is used for discretization. In total, 100 elements are used, i.e., $n = 100$. $E = 13\ MPa$, $\nu = 0.3$, and $\rho = 2000\ kg/m^3$. The half-space is truncated by using Lysmer dashpots and is modeled using `ZeroLength1D` element with `Viscous1DLinear` material. The dashpots coefficients are $c_r = \rho_r V_r B/4$ for Nodes (1), (2), (3) and (4) where $\rho_r = ho$ and $V_r = 50\ m/s$. Incoming waves are modeled by nodal forces $f_r(t) = \rho_r V_r B/4 \cdot f(t)$ applied at nodes (1), (2), (3) and (4). The function $f(t)$ is the velocity of the incident motion at depth H and is shown in Figure 124b. Rayleigh damping is used to model small strain damping with mass and stiffness proportional coefficients of 0.12442 and 0.00079, respectively. The same problem is modeled in OpenSees using Quadrilaterals with 4 nodes in plane-strain for verification. Accelerations at nodes (1) and (401) are shown in Figure 125.



(a) (b)

Figure 124: Varification for 1D site response analysis in an elastic half-space.

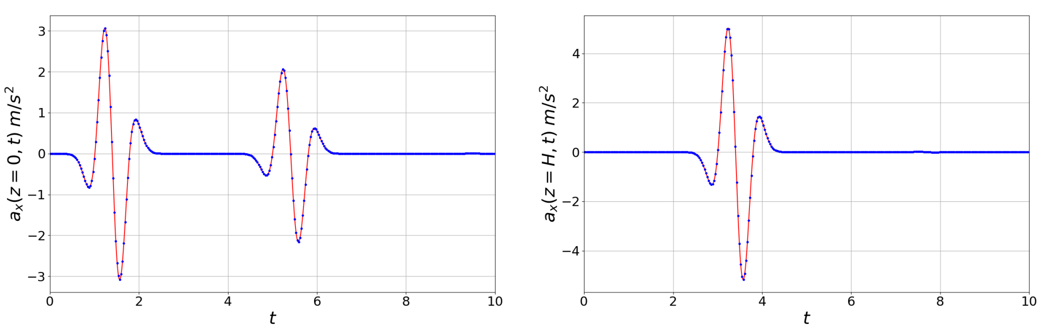


Figure 125: Accelerations at $z = H$ and $z = 0$: Opensees (. . .), SeismoVLAB (——).

The root mean square error for the displacements at node (401) is : `(5.37488e-08, 0.00000)`, while the maximum relative error for the velocity and acceleration are : `(1.01924e-07,0.00000)`, and `(9.25660e-07,0.00000)` respectively.

DEBUG CASE : J12-DY_Axial_Load_Long_Rod_PML3D

The Problem is shown in Figure 126 and is defined to test `PML3DHexa8` element with material type provided in `Elastic2DPlaneStrain`. The material has $E = 50\ MPa$, $\nu = 0.25$, and $\rho = 2000\ kg/m^3$. The rod is 100 m long and is fixed on the left hand side and isfree to move in the axial direction. Four nodal forces are placed on the right end with $P = 0.05\ MN$. The velocity responses are evaluated at the right border (blue - PC2) and PML/Hexa interface (red - PC1) and they should show no reflections after $t \geq 2$ $[s]$.

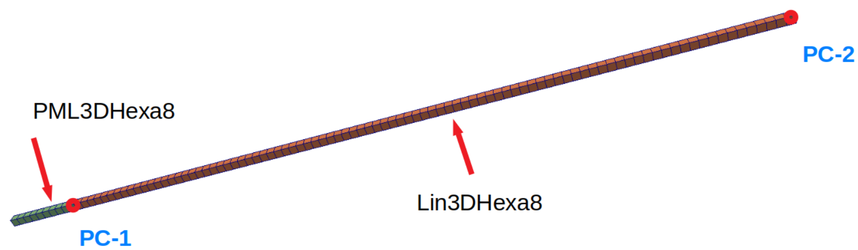


Figure 126: Verification for `PML3DHexa8` with `Elastic2DPlaneStrain` material.

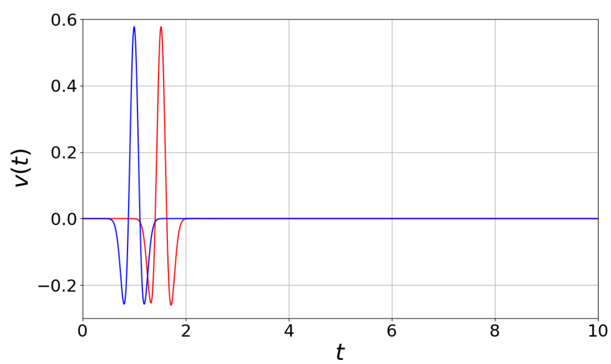


Figure 127: Nodal velocity responses in SeismoVLAB (blue PC_2) (red PC_1).

DEBUG CASE : P01-ST_DY_Lin_1D_Progressive_Mass_Elastic_ZeroLength

The problem setting is the same as shown in Figure 7 and correspond to a mass-spring-dashpot oscillator for which $M = 1$ $[kg]$, $K = 4$ $[N/m]$, $C = 0.2$ $[N\,s/m]$. The nodes (1), and (2) have the coordinate $(x, y) = (0.0, 0.0)$. Node (1) is fixed in X- and Y- directions, while node (2) is fixed in Y-direction. This problem tests the `Progressive` analysis option. First, a static analysis with $P_0 = -10$ is applied at Node (2). Then, a dynamic analysis with $P(t) = P_0 \sin(\omega_n t)$ with $\omega_n = 2$ $[rad/s]$ is applied to Node (2). The responses are verified against analytical solution. Figure 128 shows the displacement and reactive force responses at node (2).

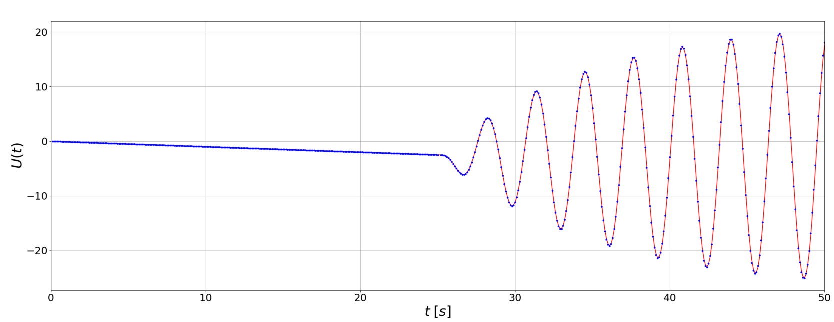


Figure 128: Nodal responses at node (2): Analytical (...), SeismoVLAB (——).

DEBUG CASE : P02-ST_kin_2D_Progressive_Moment_Elastic_Frame2

Problem setting is the same as shown in Figure 56 and is defined to test `Progressive` analysis option. For this example, all beam members have a cross-sectional area, $A = 4.0\,[in^2]$, second area moment of inertia, $I = 1.3333\,[in^4]$

and modulus of elasticity, $E = 100\,[ksi]$. The beam is $10\,[in]$ long and is discretized with 10 equal length beam elements and 11 nodes. A moment is placed at Node (11) of magnitude $M_c = 2\pi \mathrm{EI/L}\,[kip - in]$. This moment is applied in three different stages: First, a $M = M_c/3$ moment is applied, then an extra $M_c/3$ is applied, and finally the last $M_c/3$ is applied. Results should coincide with numerical solution provied by Louie L. Yaw. Figure 129 at node (11).

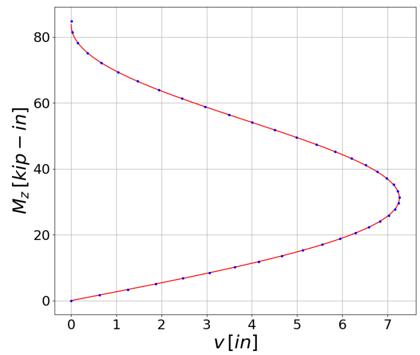


Figure 129: Moment displacement curve at (42): Analytical (...), SeismoVLAB (——).

DEBUG CASE : P03-ST_DY_Progressive_WideFlange_NonLinear_Fiber_Section_Frame2

The problem setting is the same as shown in Figure 78 and is defined to test `Progressive` analysis option. The material has modulus of elasticity $E = 29000000$, yield strength $fy = 60000$, $b = 0.01$, $a_1 = a_3 = 0.0$, $a_2 = a_4 = 1.0$, $cR_1 = 0.925$, $cR_2 = 0.15$, and a Poisson's ratio $\nu = 0.33$. The nodes (1), and (8) have the coordinate $(x, y) = (0.0, 0.0)$ and $(x, y) = (100.0, 0.0)$ respectively. Node (1) is clamped, this is fixed in X, Y-directions and rotation. First, a static axial load $P = -100000$ is applied at node (8), then a dynamic nodal force is applied at node (8) and is defined as $P(t) = 8.75 \cdot 10^5 t\,\sin(0.4\pi\,t)$. The responses are verified against OpenSees. Figure 130 shows the vertical displacement at node (2) and the moment and axial forces developed at the fixed boundary.

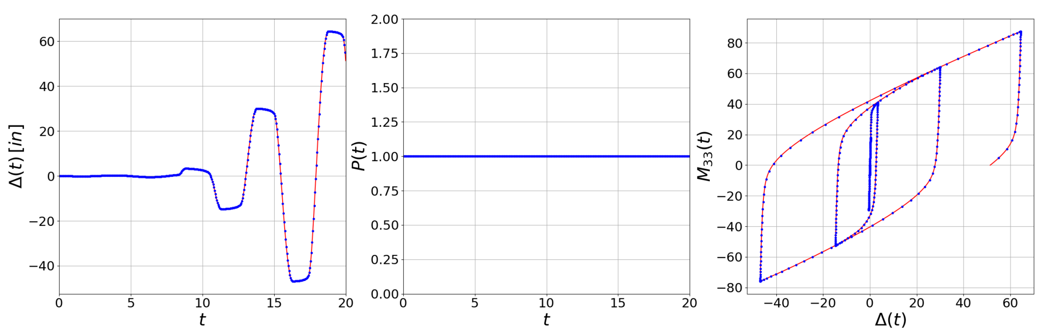


Figure 130: Nodal vertical responses at node (8) and shear force at node (8) : OpenSees (...), SeismoVLAB (——).

The root mean square error for the displacement is : `0.0198595`, while The maximum absolute difference for the displacement is : `0.0542607`.