QuakeCoRE OpenSees Training Workshop 2017

Geotechnical Analysis in OpenSees

Geotechnical Analysis

The intention of this module is to describe the types of geotechnical problems that can be simulated using OpenSees, and to discuss the essential tools that are needed to create the types of models used in geotechnical analysis.

Because most geotechnical models files are quite large in comparison to structural analyses, the current module will also present some advanced geotechnical models to demonstrate what can be done with this tool.

- OpenSees geotechnical examples can be found at (among other places)

  opensees.berkeley.edu/wiki/index.php/Examples

  opensees.berkeley.edu/wiki/index.php/FluidSolidPorousMaterial

  opensees.berkeley.edu/wiki/index.php/PressureDependMultiYield_Material

  opensees.berkeley.edu/wiki/index.php/PressureIndependMultiYield_Material

Geotechnical Analysis

QuakeCoRE
NZ Centre for Earthquake Resilience

Types of geotechnical problems that can be solved with OpenSees:

- Static Problems:
    - Deformation analyses (1D, 2D, 3D)
    - Consolidation problems (diffusion problems)
    - Soil-structure interaction problems
        - Shallow foundation (bearing capacity, settlement)
        - Deep foundations (vertical and lateral capacity)

- Dynamic Problems
    - Free-field analysis (site response)
    - Liquefaction and liquefaction-induced phenomena
    - Soil-structure interaction problems

Geotechnical Analysis

QuakeCoRE
NZ Centre for Earthquake Resilience

What tools will we need?

- 2D and 3D solid elements to characterize the soil domain (continuum)
- Appropriate boundary conditions to accurately represent the soil domain boundaries and not significantly affect the area of interest
- Robust constitutive models to characterize the stress-strain response of the soil under monotonic and cyclic loading conditions
- Interface elements and materials to capture the interaction between the soil and any adjacent structures
- Everything else we are discussing in this workshop (how to create beam elements, apply loads, boundary conditions, record results, perform analyses, etc)

Finite Element Tools

QuakeCoRE
NZ Centre for Earthquake Resilience

Single-phase formulations

- To capture the response of dry soils (or total stress analysis)
- Only need a single phase
  - Phase 1: soil skeleton

Multi-phase formulations

- To capture the response of saturated soils (effective stress analysis)
- Now need two phases
  - Phase 1: soil skeleton
  - Phase 2: pore water

Zero-Length elements

- To capture interface response between solid and beam elements, and to apply absorbent boundary conditions

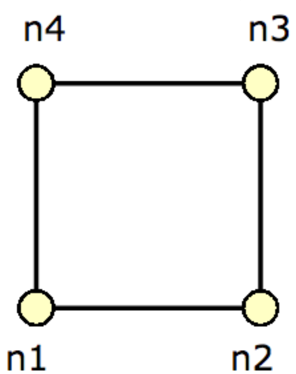
Contact elements

- To capture interface response between different bodies

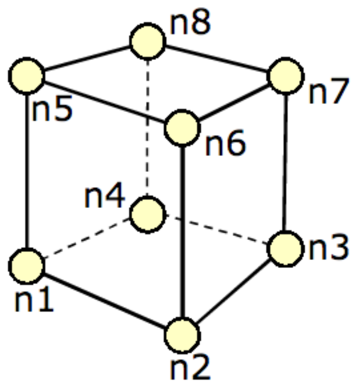
Single Phase Formulations

Small deformation solid elements

- 2D quadrilateral elements (quads) – typically with 4 nodes
- 3D hexahedral elements (bricks) – typically with 8 nodes



quad (4 node)



stdBrick (8 node)

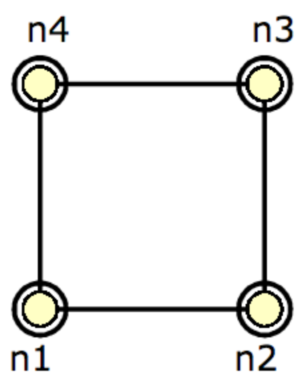
Multi-Phase Formulations

Fully coupled **u**-p elements

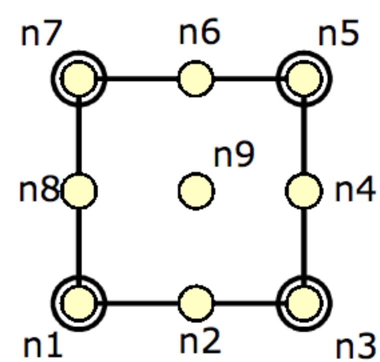
- 2D quadrilateral elements (quads) – 4 and 9 nodes
- 3D hexahedral elements (bricks) – 8 and 20 nodes

Degrees of freedom are:

- Solid displacements **u** on
- Pore fluid pressure p on



quadUP



9_4_quadUP

Constitutive Models

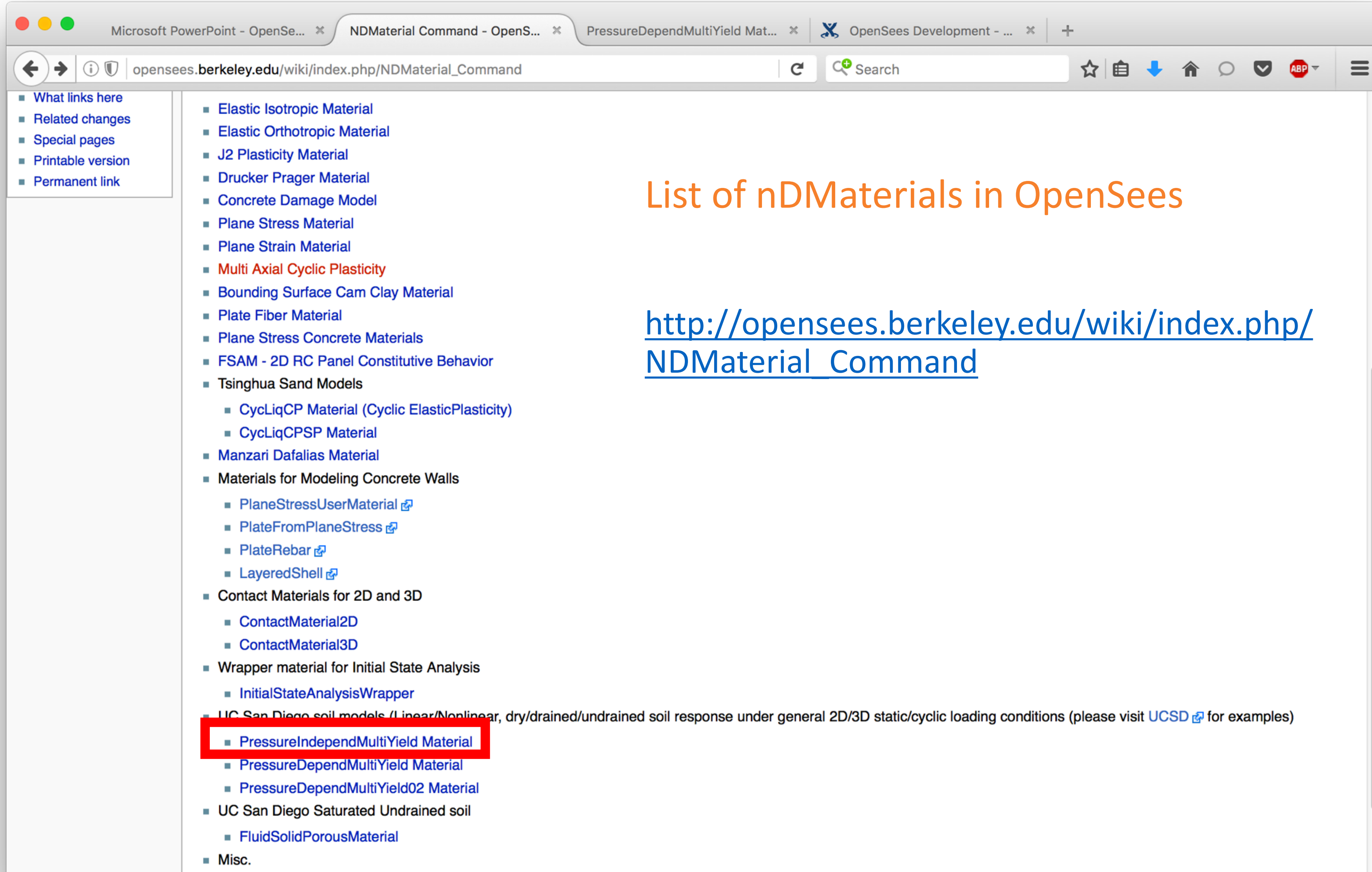
QuakeCoRE
NZ Centre for Earthquake Resilience

In 2D and 3D, we need to use material models that can capture multi-dimensional response

- For this reason nDMaterial objects will be our primary tools for geotechnical modeling in OpenSees
- There are many available options
  - General nDMaterials (e.g. ElasticIsotropic, J2Plasticity)
  - Soil nDMaterials (for both total and effective stress analysis)
  - nDMaterials for modeling concrete walls
- We will also need to make use of uniaxialMaterials for things like implementing absorbing boundaries and soil-structure interaction problems.

Constitutive Models

QuakeCoRE
NZ Centre for Earthquake Resilience



List of nDMaterials in OpenSees

http://opensees.berkeley.edu/wiki/index.php/NDMaterial_Command

Constitutive Models

PressureIndependMultiYield Material

Command_Manual

PressureIndependMultiYield material is an elastic-plastic material in which plasticity exhibits only in the deviatoric stress-strain response. The volumetric stress-strain response is linear-elastic and is independent of the deviatoric response. This material is implemented to simulate monotonic or cyclic response of materials whose shear behavior is insensitive to the confinement change. Such materials include, for example, organic soils or clay under fast (undrained) loading conditions.

During the application of gravity load (and static loads if any), material behavior is linear elastic. In the subsequent dynamic (fast) loading phase(s), the stress-strain response is elastic-plastic (see MATERIAL STAGE UPDATE below). Plasticity is formulated based on the multi-surface (nested surfaces) concept, with an associative flow rule. The yield surfaces are of the Von Mises type.

OUTPUT INTERFACE:

The following information may be extracted for this material at a given integration point, using the OpenSees Element Recorder facility (McKenna and Fenves 2001)®: "**stress**", "**strain**", "**backbone**", or "**tangent**".

For 2D problems, the stress output follows this order: σ_{xx}, σ_{yy}, σ_{zz}, σ_{xy}, η_r, where η_r is the ratio between the shear (deviatoric) stress and peak shear strength at the current confinement ($0<=\eta_r<=1.0$). The strain output follows this order: ε_{xx}, ε_{yy}, γ_{xy}.

For 3D problems, the stress output follows this order: σ_{xx}, σ_{yy}, σ_{zz}, σ_{xy}, σ_{yz}, σ_{zx}, η_r, and the strain output follows this order: ε_{xx}, ε_{yy}, ε_{zz}, γ_{xy}, γ_{yz}, γ_{zx}.

The "**backbone**" option records (secant) shear modulus reduction curves at one or more given confinements. The specific recorder command is as follows:

recorder Element –ele $eleNum -file $fName -dT $deltaT material $GaussNum backbone $p1 <$p2 …>

where p1, p2, … are the confinements at which modulus reduction curves are recorded. In the output file, corresponding to each given confinement there are two columns: shear strain γ and secant modulus G_s. The number of rows equals the number of yield surfaces.

nDmaterial PressureIndependMultiYield $tag $nd $rho $refShearModul $refBulkModul $cohesi $peakShearStra <$frictionAng=0. $refPress=100. $pressDependCoe=0. $noYieldSurf=20 <$r1 $Gs1 …> >

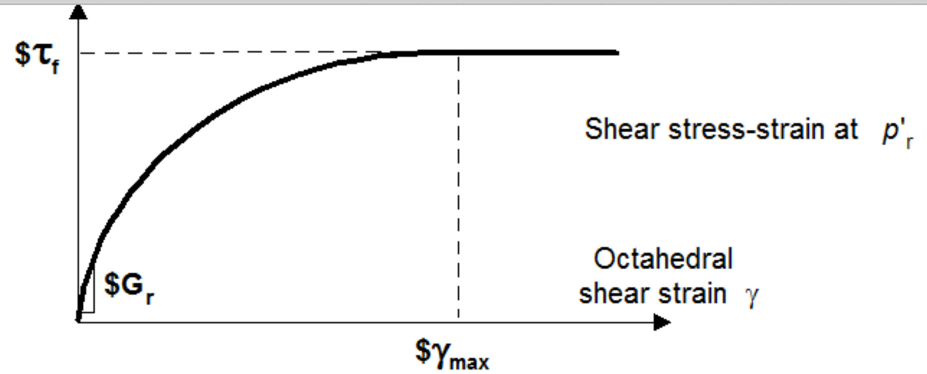
navigation
- OpenSees Home
- Message Board
- OpenSees Wiki
- User Documentation
- Download Page
- Source Code

search
Go | Search

toolbox
- What links here
- Related changes
- Special pages
- Printable version
- Permanent link

Constitutive Models

opensees.berkeley.edu/wiki/index.php/PressureIndependMultiYield_Material



$tag	A positive integer uniquely identifying the material among all nDMaterials.
$nd	Number of dimensions, 2 for plane-strain, and 3 for 3D analysis.
$rho	Saturated soil mass density.
$refShearModul ($G_r$)	Reference low-strain shear modulus, specified at a reference mean effective confining pressure refPress of p'_r (see below).
$refBulkModul ($B_r$)	Reference bulk modulus, specified at a reference mean effective confining pressure refPress of p'_r (see below).
$cohesi (c)	Apparent cohesion at zero effective confinement.
$peakShearStra ($\gamma_{max}$)	An octahedral shear strain at which the maximum shear strength is reached, specified at a reference mean effective confining pressure refPress of p'_r (see below).
$frictionAng (Φ)	Friction angle at peak shear strength in degrees, optional (default is 0.0).
$refPress ($p'_r$)	Reference mean effective confining pressure at which Gr, Br, and γ_{max} are defined, optional (default is 100. kPa).
$pressDependCoe (d)	A positive constant defining variations of G and B as a function of instantaneous effective confinement p'(default is 0.0):: $G = G_r\left(\frac{p'}{p'_r}\right)^d$ $B = B_r\left(\frac{p'}{p'_r}\right)^d$ **If Φ=0, d is reset to 0.0.**
$noYieldSurf	Number of yield surfaces, optional (must be less than 40, default is 20). The surfaces are generated based on the hyperbolic relation defined in Note 2 below.
$r, $Gs	Instead of automatic surfaces generation (Note 2), **you can define yield surfaces directly based on desired shear modulus reduction curve.** To do so, add a minus sign in front of noYieldSurf, then provide noYieldSurf pairs of shear strain (γ) and modulus ratio (G_s) values. For example, to define 10 surfaces: … $-10\gamma_1 G_{s1}$ … $\gamma_{10} G_{s10}$ … See Note 3 below for some important notes.

Constitutive Models

QuakeCoRE
NZ Centre for Earthquake Resilience

opensees.berkeley.edu/wiki/index.php/PressureIndependMultiYield_Material

4. SUGGESTED PARAMETER VALUES

For user convenience, a table is provided below as a quick reference for selecting parameter values. However, use of this table should be of great caution, and other information should be incorporated wherever possible.

Parameters	Soft Clay	Medium Clay	Stiff Clay
rho	1.3 ton/m^3 or 1.217x10^{-4} (lbf)(s^2)/in^4	1.5 ton/m^3 or 1.404x10^{-4} (lbf)(s^2)/in^4	1.8 ton/m^3 or 1.685x10^{-4} (lbf)(s^2)/in^4
refShearModul	1.3x10^4 kPa or 1.885x10^3 psi	6.0x10^4 kPa or 8.702x10^4 psi	1.5x10^5 kPa or 2.176x10^4 psi
refBulkModu	6.5x10^4 kPa or 9.427x10^3 psi	3.0x10^5 kPa or 4.351x10^4 psi	7.5x10^5 kPa or 1.088x10^5 psi
cohesi	18 kPa or 2.611 psi	37 kPa or 5.366 psi	75 kPa or 10.878 psi
peakShearStra (at p'_r=80 kPa or 11.6 psi)	0.1	0.1	0.1
frictionAng	0	0	0
pressDependCoe	0	0	0

Pressure Independent Material Examples:

Material in elastic state	
Example 1	Single 2D plane-strain quadrilateral element, subjected to sinusoidal base shaking
Example 2	Single 2D quadrilateral element, subjected to monotonic pushover (English units version)

Code Developed by: UC San Diego (Dr. Zhaohui Yang):

UC San Diego Soil Model:

- NDMaterial Command
  - UC San Diego soil models (Linear/Nonlinear, dry/drained/undrained soil response under general 2D/3D static/cyclic loading conditions (please visit UCSD for examples)
    - **PressureIndependMultiYield Material**
    - PressureDependMultiYield Material

Constitutive Models

QuakeCoRE
NZ Centre for Earthquake Resilience

For geotechnical analysis, it is critical that we achieve the proper initial state of stress in the soil before we start applying external loads

- This is typically done using a staged analysis where settings and/or properties of the soil materials are changed from the model file after completing different analysis stages
- Some additional commands that make this possible:

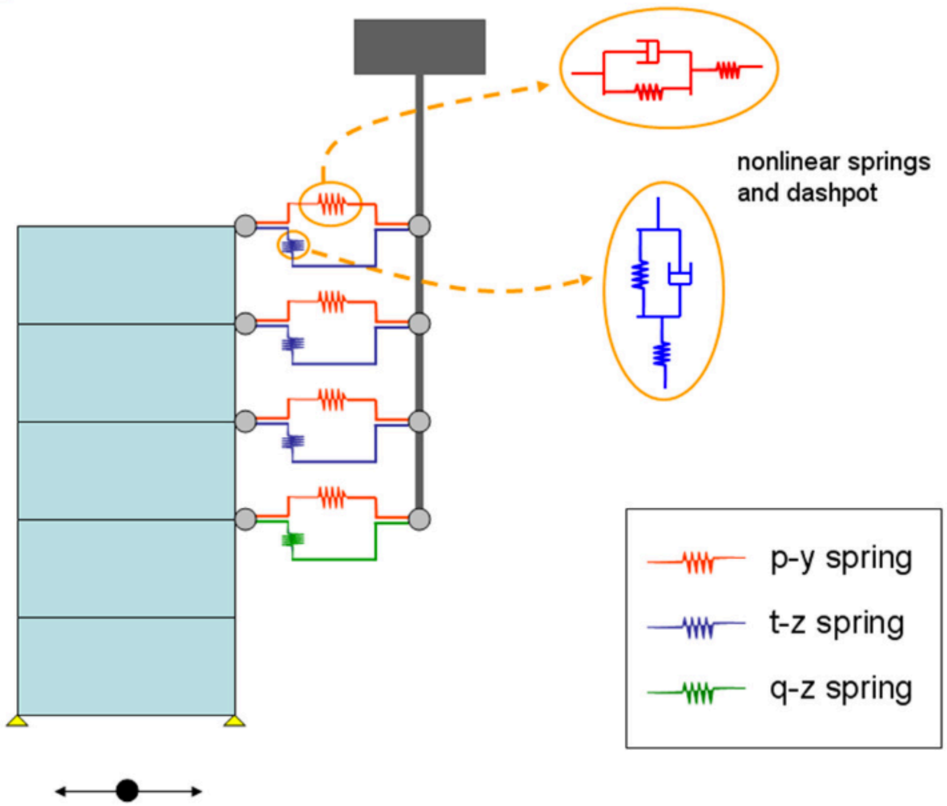
  updateMaterialStage –material $matTag –stage $stageNum

  setParameter –value $pVal –eleRange $e1 $eN $paramName

- Documentation will discuss what options are available for a given nDMaterial object

Constitutive Models

For soil-pile interaction problems, we need a way to capture the interface response between solid elements (soil) and beam elements (pile)



We can use Py, Tz, and Qz uniaxialMaterials and zeroLength elements for this purpose.

- PySimple1
- TzSimple1
- QzSimple1

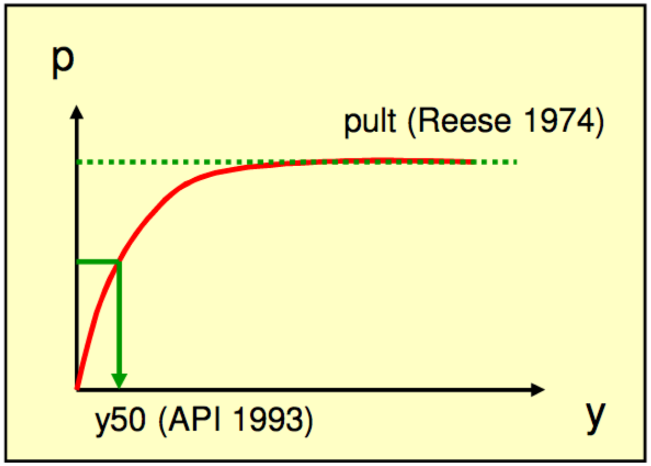
- PyLiq1
- TzLiq1

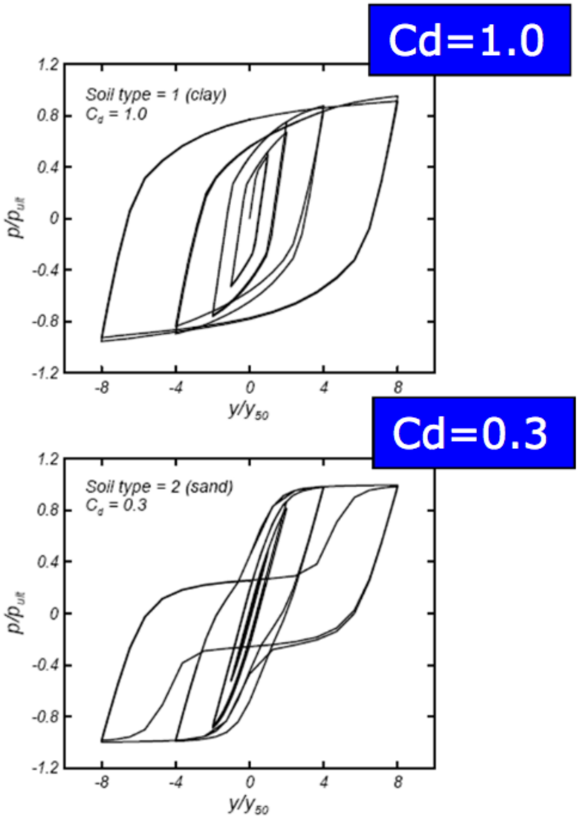
Constitutive Models

PySimple uniaxialMaterial for modeling lateral soil-pile interaction response

uniaxialMaterial PySimple1 $tag $soilType $pult $y50 $Cd <$c>

$soilType → 1 = clay (Matlock), 2 = sand (API)
$pult → ultimate capacity of p-y material
$y50 → displacement at 50% of pult
$Cd → drag resistance (1 = no gap, <1 = gap)
$c → viscous damping





Constitutive Models

QuakeCoRE
NZ Centre for Earthquake Resilience

PyLiq1 uniaxialMaterial for lateral soil-pile interaction response with consideration for strength reduction with build-up of excess pore pressure

uniaxialMaterial PyLiq1 $tag $soilType $pult $y50 $Cd $c $pRes $solidElem1 $solidElem2

$soilType → 1 = clay (Matlock), 2 = sand (API)

$pult → ultimate capacity of p-y material

$y50 → displacement at 50% of pult

$Cd → drag resistance (1 = no gap, <1 = gap)

$c → viscous damping

$pRes → residual p-y resistance for r_u = 1.0

$solidElem1 and $solidElem2 → solid elements from which the PyLiq1 object will obtain mean effective stresses and pore pressures

Constitutive Models

PyLiq1 uniaxialMaterial for lateral soil-pile interaction response with consideration for strength reduction with build-up of excess pore pressure

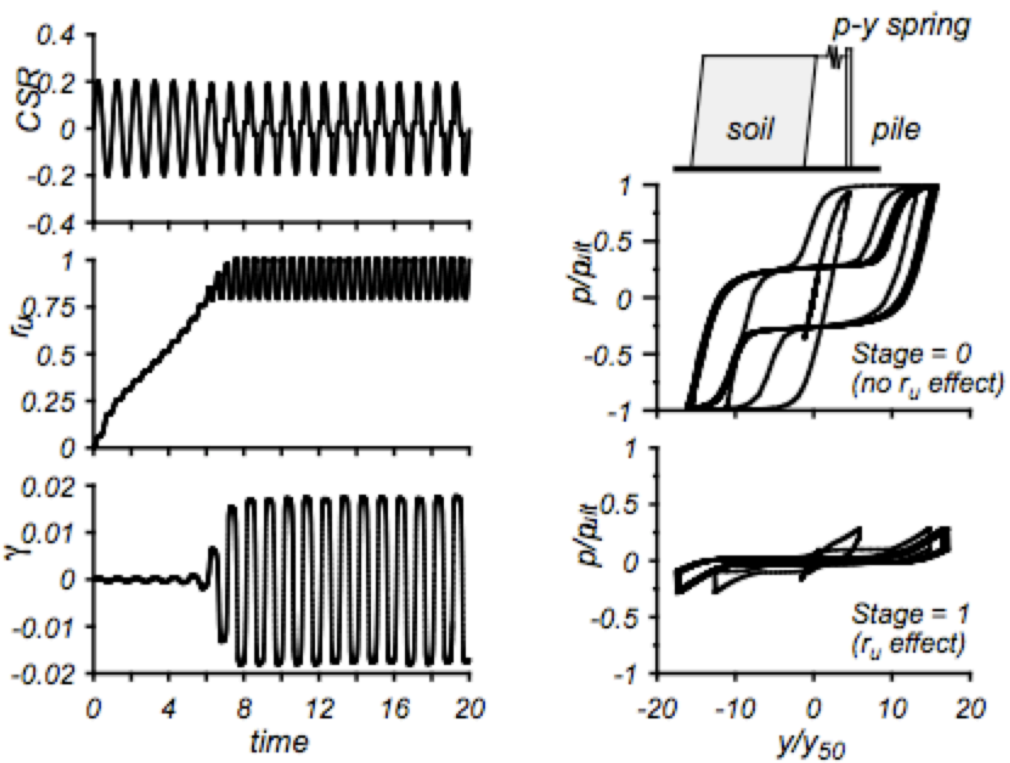


FIG. Example of PyLiq1 behavior during liquefaction without lateral spreading.

Boundary Conditions

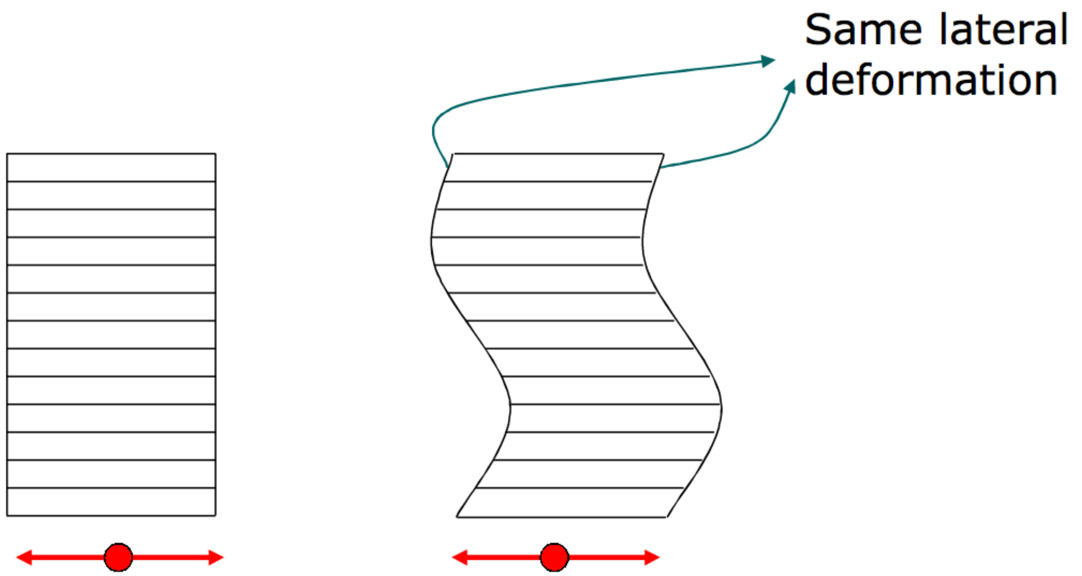
Periodic boundary conditions can be implemented using the equalDOF command (which we have seen before)

equalDOF $retainedNode $constrainedNode $dof1 $dof2 ...

$retainedNode → tag of the retained node (master node)

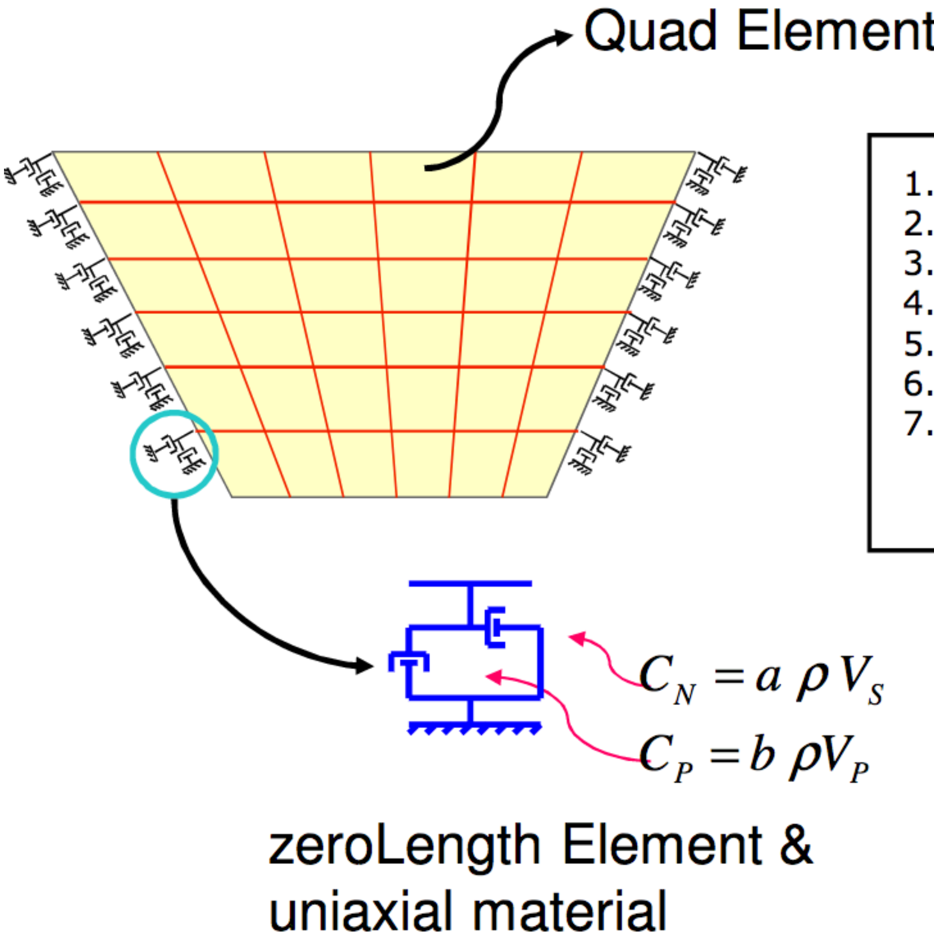
$constrainedNode → tag of the constrained node (slave node)

$dof1 $dof2 ... → constrained dof



Boundary Conditions

Absorbing boundaries can also be implemented using tools and techniques that we've seen already




```
1. set DampP 755
2. set DampN 1216
3. uniaxialMaterial Elastic  1  0 $DampP
4. uniaxialMaterial Elastic  2  0 $DampN
5. node 1 16.0 0.0
6. node 2 16.0 0.0
7. element zeroLength 1 1  2  -mat 1  2
     -dir 1 2 –orient 1 –2 0 2 1 0
```

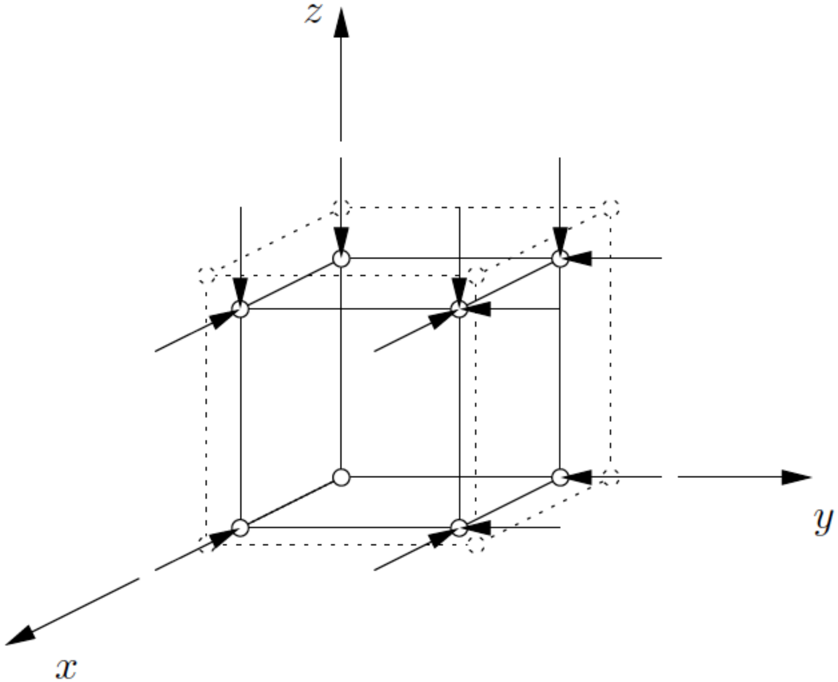
QuakeCoRE
NZ Centre for Earthquake Resilience

Geotechnical and Soil Mechanics Example Analyses

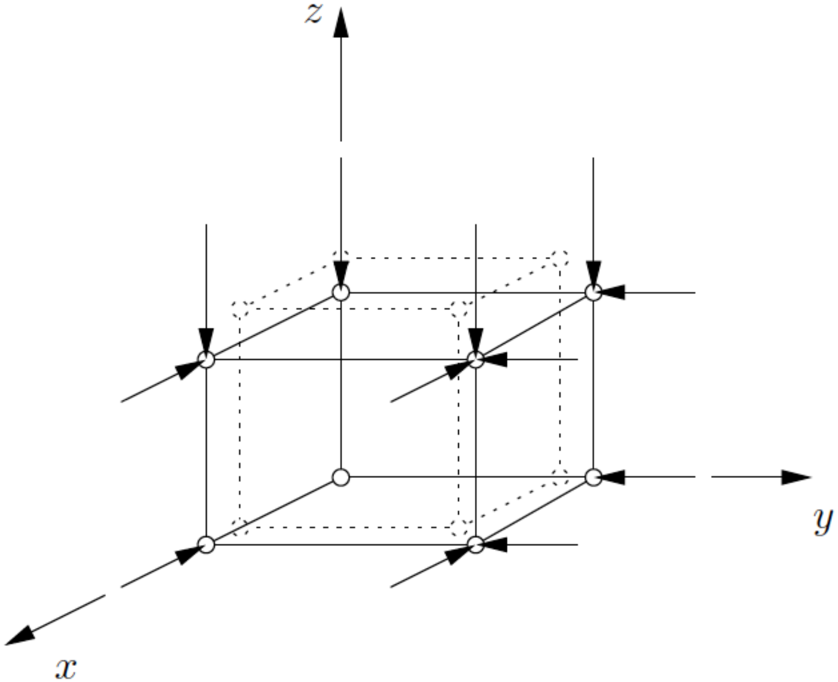
Single-Element Testing

QuakeCoRE
NZ Centre for Earthquake Resilience

It is often useful to test, verify, and calibrate soil constitutive models (nDMaterials) using single element tests that put materials through the same stress paths as laboratory tests for soils.



(a)



(b)

For example, a conventional triaxial compression (CTC) test can be modeled using a single element as shown in this schematic.

Single Element Testing

QuakeCoRE
NZ Centre for Earthquake Resilience

ctcTest.tcl (~/QuakeCoRE/OpenSe...dule3_geotechnica

```
wipe

# Confinement Stress
set pConf -50.0
# Deviatoric strain
set devDisp -0.2
# Permeablity
set perm 1.0e-9

# Rayleigh damping parameters
set damp   0.1
set omega1 0.0157
set omega2 64.123
set a1 [expr 2.0*$damp/($omega1+$omega2)]
set a0 [expr $a1*$omega1*$omega2]

# Create a 3D model with 4 Degrees of Freedom
model BasicBuilder -ndm 3 -ndf 4

# Create nodes
node 1  1.0 0.0 0.0
node 2  1.0 1.0 0.0
node 3  0.0 1.0 0.0
node 4  0.0 0.0 0.0
node 5  1.0 0.0 1.0
node 6  1.0 1.0 1.0
node 7  0.0 1.0 1.0
node 8  0.0 0.0 1.0

# Create Fixities
fix 1   0 1 1 1
fix 2   0 0 1 1
fix 3   1 0 1 1
fix 4   1 1 1 1
fix 5   0 1 0 1
fix 6   0 0 0 1
fix 7   1 0 0 1
fix 8   1 1 0 1
```

We'll quickly walk through the single element CTC test file. The first task is to define some general terms that will be used in the analysis.

We define the desired confining stress, the applied deviatoric displacement, and the permeability of the soil (for undrained conditions, set this to something very small)

We also define the Rayleigh damping parameters at the beginning of the file so they are easy to find if we want to change them. The Rayleigh command is called later, once we've defined the domain.

Single Element Testing

ctcTest.tcl (~/QuakeCoRE/OpenSe...dule3_geotechnica

```
wipe

# Confinement Stress
set pConf -50.0
# Deviatoric strain
set devDisp -0.2
# Permeablity
set perm 1.0e-9

# Rayleigh damping parameters
set damp   0.1
set omega1 0.0157
set omega2 64.123
set a1 [expr 2.0*$damp/($omega1+$omega2)]
set a0 [expr $a1*$omega1*$omega2]

# Create a 3D model with 4 Degrees of Freedom
model BasicBuilder -ndm 3 -ndf 4

# Create nodes
node 1  1.0 0.0 0.0
node 2  1.0 1.0 0.0
node 3  0.0 1.0 0.0
node 4  0.0 0.0 0.0
node 5  1.0 0.0 1.0
node 6  1.0 1.0 1.0
node 7  0.0 1.0 1.0
node 8  0.0 0.0 1.0

# Create Fixities
fix 1   0 1 1 1
fix 2   0 0 1 1
fix 3   1 0 1 1
fix 4   1 1 1 1
fix 5   0 1 0 1
fix 6   0 0 0 1
fix 7   1 0 0 1
fix 8   1 1 0 1
```

We want our analysis to start from a truly hydrostatic state of stress, so we need to use a 3D brick element and define the domain with 3 dimensions (-ndm 3).

We also want the test to be undrained, so we need to use a u-p element and a small permeability. This requires us to use a domain with 4 dof (-ndf 4) as the pore pressure is assigned to the 4th dof.

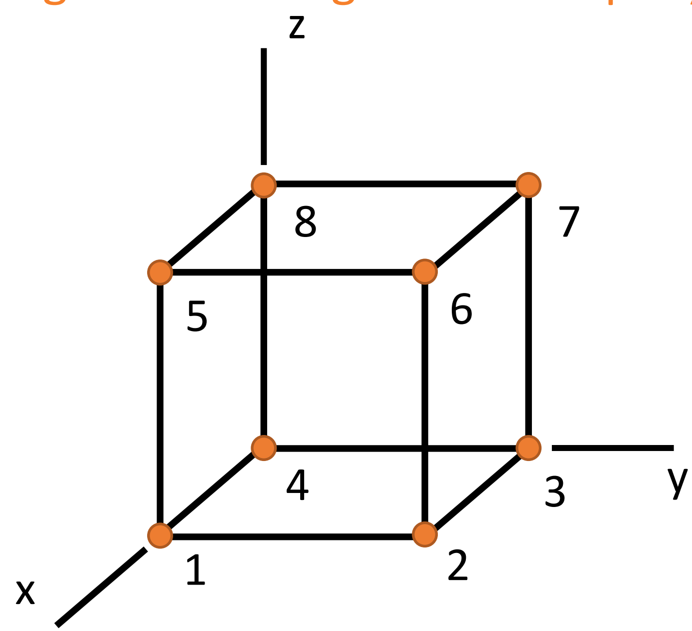
Single Element Testing

ctcTest.tcl (~/QuakeCoRE/OpenSe...dule3_geotechnica

```
wipe

# Confinement Stress
set pConf -50.0
# Deviatoric strain
set devDisp -0.2
# Permeablity
set perm 1.0e-9

# Rayleigh damping parameters
set damp   0.1
set omega1 0.0157
set omega2 64.123
set a1 [expr 2.0*$damp/($omega1+$omega2)]
set a0 [expr $a1*$omega1*$omega2]

# Create a 3D model with 4 Degrees of Freedom
model BasicBuilder -ndm 3 -ndf 4

# Create nodes
node 1  1.0 0.0 0.0
node 2  1.0 1.0 0.0
node 3  0.0 1.0 0.0
node 4  0.0 0.0 0.0
node 5  1.0 0.0 1.0
node 6  1.0 1.0 1.0
node 7  0.0 1.0 1.0
node 8  0.0 0.0 1.0

# Create Fixities
fix 1   0 1 1 1
fix 2   0 0 1 1
fix 3   1 0 1 1
fix 4   1 1 1 1
fix 5   0 1 0 1
fix 6   0 0 0 1
fix 7   1 0 0 1
fix 8   1 1 0 1
```

The nodes are created in 3D (requires 3 coordinates per node). The size of the element doesn't really matter for this analysis, a 1.0 unit cube is used.

During the initial analysis phase, we fix the pore pressure dof (dof 4) such that pressure cannot develop (analogous to having the valve open).



Single Element Testing

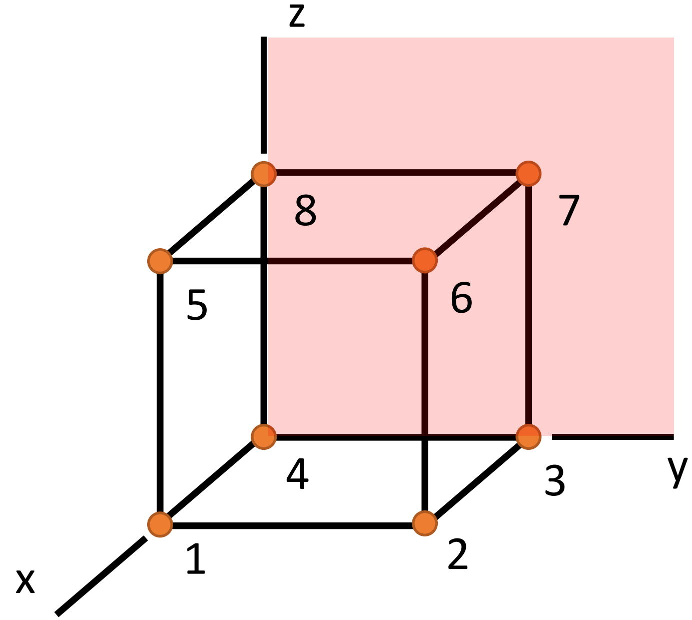
QuakeCoRE
NZ Centre for Earthquake Resilience

ctcTest.tcl (~/QuakeCoRE/OpenSe...dule3_geotechnica

```
wipe

# Confinement Stress
set pConf -50.0
# Deviatoric strain
set devDisp -0.2
# Permeablity
set perm 1.0e-9

# Rayleigh damping parameters
set damp   0.1
set omega1 0.0157
set omega2 64.123
set a1 [expr 2.0*$damp/($omega1+$omega2)]
set a0 [expr $a1*$omega1*$omega2]

# Create a 3D model with 4 Degrees of Freedom
model BasicBuilder -ndm 3 -ndf 4

# Create nodes
node 1  1.0 0.0 0.0
node 2  1.0 1.0 0.0
node 3  0.0 1.0 0.0
node 4  0.0 0.0 0.0
node 5  1.0 0.0 1.0
node 6  1.0 1.0 1.0
node 7  0.0 1.0 1.0
node 8  0.0 0.0 1.0

# Create Fixities
fix 1   0 1 1 1
fix 2   0 0 1 1
fix 3   1 0 1 1
fix 4   1 1 1 1
fix 5   0 1 0 1
fix 6   0 0 0 1
fix 7   1 0 0 1
fix 8   1 1 0 1
```

The nodes are created in 3D (requires 3 coordinates per node). The size of the element doesn't really matter for this analysis, a 1.0 unit cube is used.

During the initial analysis phase, we fix the pore pressure dof (dof 4) such that pressure cannot develop (analogous to having the valve open).



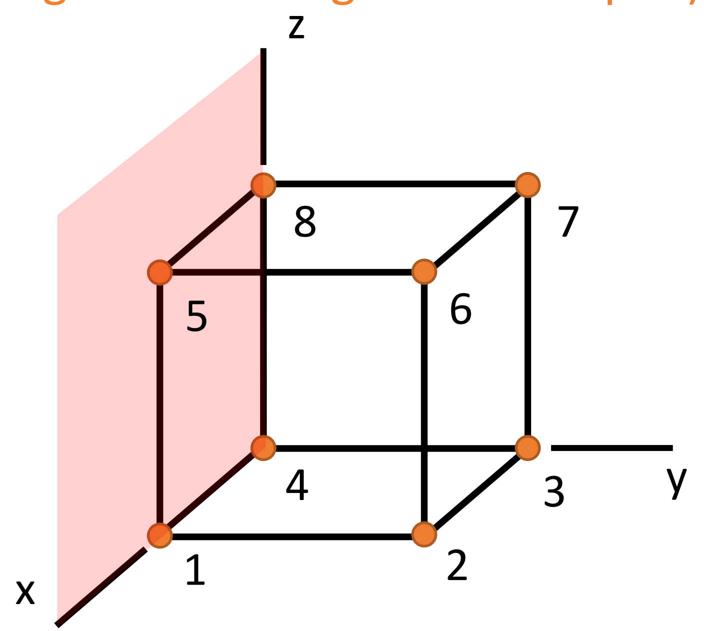
Single Element Testing

ctcTest.tcl (~/QuakeCoRE/OpenSe...dule3_geotechnica

```
wipe

# Confinement Stress
set pConf -50.0
# Deviatoric strain
set devDisp -0.2
# Permeablity
set perm 1.0e-9

# Rayleigh damping parameters
set damp   0.1
set omega1 0.0157
set omega2 64.123
set a1 [expr 2.0*$damp/($omega1+$omega2)]
set a0 [expr $a1*$omega1*$omega2]

# Create a 3D model with 4 Degrees of Freedom
model BasicBuilder -ndm 3 -ndf 4

# Create nodes
node 1  1.0 0.0 0.0
node 2  1.0 1.0 0.0
node 3  0.0 1.0 0.0
node 4  0.0 0.0 0.0
node 5  1.0 0.0 1.0
node 6  1.0 1.0 1.0
node 7  0.0 1.0 1.0
node 8  0.0 0.0 1.0

# Create Fixities
fix 1   0 1 1 1
fix 2   0 0 1 1
fix 3   1 0 1 1
fix 4   1 1 1 1
fix 5   0 1 0 1
fix 6   0 0 0 1
fix 7   1 0 0 1
fix 8   1 1 0 1
```

The nodes are created in 3D (requires 3 coordinates per node). The size of the element doesn't really matter for this analysis, a 1.0 unit cube is used.

During the initial analysis phase, we fix the pore pressure dof (dof 4) such that pressure cannot develop (analogous to having the valve open).



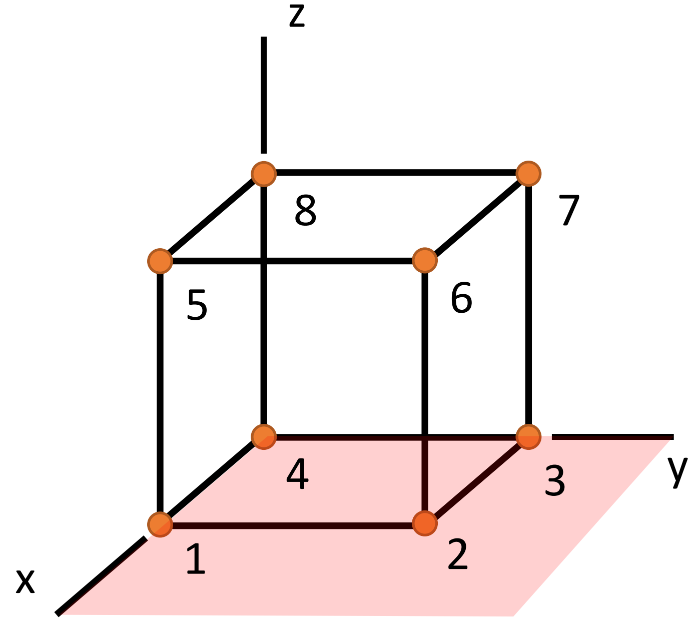
Single Element Testing

ctcTest.tcl (~/QuakeCoRE/OpenSe...dule3_geotechnica

```
wipe

# Confinement Stress
set pConf -50.0
# Deviatoric strain
set devDisp -0.2
# Permeablity
set perm 1.0e-9

# Rayleigh damping parameters
set damp   0.1
set omega1 0.0157
set omega2 64.123
set a1 [expr 2.0*$damp/($omega1+$omega2)]
set a0 [expr $a1*$omega1*$omega2]

# Create a 3D model with 4 Degrees of Freedom
model BasicBuilder -ndm 3 -ndf 4

# Create nodes
node 1  1.0 0.0 0.0
node 2  1.0 1.0 0.0
node 3  0.0 1.0 0.0
node 4  0.0 0.0 0.0
node 5  1.0 0.0 1.0
node 6  1.0 1.0 1.0
node 7  0.0 1.0 1.0
node 8  0.0 0.0 1.0

# Create Fixities
fix 1   0 1 1 1
fix 2   0 0 1 1
fix 3   1 0 1 1
fix 4   1 1 1 1
fix 5   0 1 0 1
fix 6   0 0 0 1
fix 7   1 0 0 1
fix 8   1 1 0 1
```

The nodes are created in 3D (requires 3 coordinates per node). The size of the element doesn't really matter for this analysis, a 1.0 unit cube is used.

During the initial analysis phase, we fix the pore pressure dof (dof 4) such that pressure cannot develop (analogous to having the valve open).



Single Element Testing

QuakeCoRE
NZ Centre for Earthquake Resilience

ctcTest.tcl (~/QuakeCoRE/OpenSe...dule3_geotechnicalExamples) - VIM

```
# define material and properties
set mTag  1
set mDen  1.8
set fang  31.4
set ptang 26.5
set eNot  0.7
set E     90000.0
set nu    0.40
set G     [expr $E/(2.0*(1.0+$nu))]
set B     [expr $E/(3.0*(1.0-2.0*$nu))]
nDMaterial PressureDependMultiYield $mTag 3 $mDen $G $B $fang 0.1 80 0.5 \
                                        $ptang 0.17 0.4 10  10 0.015 1.0

# Create element
set fBulk 2.2e6
set fDen  1.0
set alpha 1.5e-5
element SSPbrickUP 1 1 2 3 4 5 6 7 8 $mTag $fBulk $fDen $perm $perm $perm \
                                     $eNot $alpha

# Create recorders
recorder Node    -file disp.out   -time -nodeRange 5 8 -dof 3 disp
recorder Node    -file press.out  -time -nodeRange 1 8 -dof 4 vel
recorder Element -file stress.out -time stress
recorder Element -file strain.out -time strain

# Create analysis
constraints Penalty 1.0e17 1.0e17
test        NormDispIncr 1.0e-4 20 1
algorithm   Newton
numberer    RCM
system      BandGeneral
integrator  Newmark 0.5 0.25
rayleigh    $a0 0.0 $a1 0.0
analysis    Transient

# Apply confinement pressure
```

The PressureDependMultiYield nDMaterial is used here. The inputs and capabilities of this material model are discussed at:

http://opensees.berkeley.edu/wiki/index.php/PressureDependMultiYield_Material

Single Element Testing

QuakeCoRE
NZ Centre for Earthquake Resilience

ctcTest.tcl (~/QuakeCoRE/OpenSe...dule3_geotechnicalExamples) - VIM

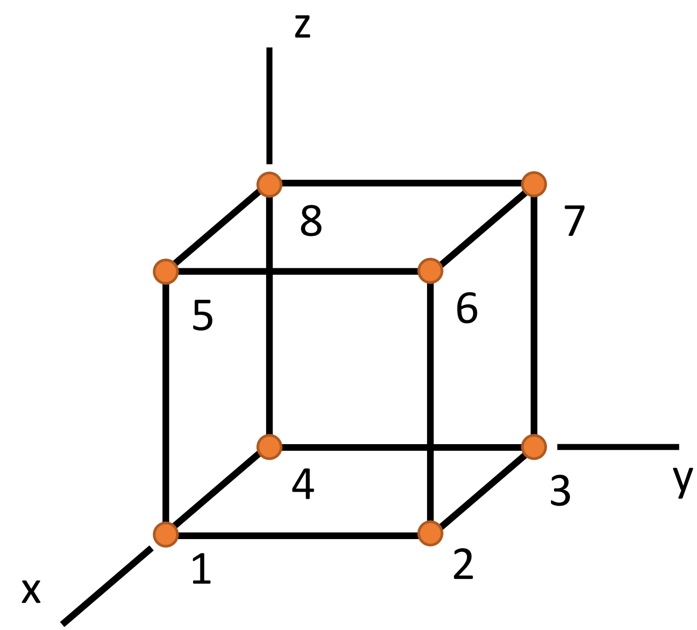
```
# define material and properties
set mTag  1
set mDen  1.8
set fang  31.4
set ptang 26.5
set eNot  0.7
set E     90000.0
set nu    0.40
set G     [expr $E/(2.0*(1.0+$nu))]
set B     [expr $E/(3.0*(1.0-2.0*$nu))]
nDMaterial PressureDependMultiYield $mTag 3 $mDen $G $B $fang 0.1 80 0.5 \
                                        $ptang 0.17 0.4 10  10 0.015 1.0

# Create element
set fBulk 2.2e6
set fDen  1.0
set alpha 1.5e-5
element SSPbrickUP 1 1 2 3 4 5 6 7 8 $mTag $fBulk $fDen $perm $perm $perm \
                                     $eNot $alpha

# Create recorders
recorder Node    -file disp.out   -time -nodeRange 5 8 -dof 3 disp
recorder Node    -file press.out  -time -nodeRange 1 8 -dof 4 vel
recorder Element -file stress.out -time stress
recorder Element -file strain.out -time strain

# Create analysis
constraints Penalty 1.0e17 1.0e17
test        NormDispIncr 1.0e-4 20 1
algorithm   Newton
numberer    RCM
system      BandGeneral
integrator  Newmark 0.5 0.25
rayleigh    $a0 0.0 $a1 0.0
analysis    Transient

# Apply confinement pressure
```

The SSPbrickUP element is used. It requires some information about the pore fluid (water in this case) and the initial void ratio of the soil.

http://opensees.berkeley.edu/wiki/index.php/SSPbrickUP_Element

Single Element Testing

```
ctcTest.tcl (~/QuakeCoRE/OpenSe...dule3_geotechnicalExamples

# define material and properties
set mTag  1
set mDen  1.8
set fang  31.4
set ptang 26.5
set eNot  0.7
set E     90000.0
set nu    0.40
set G     [expr $E/(2.0*(1.0+$nu))]
set B     [expr $E/(3.0*(1.0-2.0*$nu))]
nDMaterial PressureDependMultiYield $mTag 3 $mDen $G $B $fang 0.1 80
                                        $ptang 0.17 0.4 10  10 0.015

# Create element
set fBulk 2.2e6
set fDen  1.0
set alpha 1.5e-5
element SSPbrickUP 1 1 2 3 4 5 6 7 8 $mTag $fBulk $fDen $perm $perm
                                         $eNot $alpha

# Create recorders
recorder Node    -file disp.out   -time -nodeRange 5 8 -dof 3 disp
recorder Node    -file press.out  -time -nodeRange 1 8 -dof 4 vel
recorder Element -file stress.out -time stress
recorder Element -file strain.out -time strain

# Create analysis
constraints Penalty 1.0e17 1.0e17
test        NormDispIncr 1.0e-4 20 1
algorithm   Newton
numberer    RCM
system      BandGeneral
integrator  Newmark 0.5 0.25
rayleigh    $a0 0.0 $a1 0.0
analysis    Transient

# Apply confinement pressure
```



Note how the element connectivity is specified in a counterclockwise order. This is not the only node ordering we could have used, but the node numbers were chosen to make it easy to input into the element.

Single Element Testing

QuakeCoRE
NZ Centre for Earthquake Resilience

ctcTest.tcl (~/QuakeCoRE/OpenSe...dule3_geotechnicalExamples) - VIM

```
# define material and properties
set mTag  1
set mDen  1.8
set fang  31.4
set ptang 26.5
set eNot  0.7
set E     90000.0
set nu    0.40
set G     [expr $E/(2.0*(1.0+$nu))]
set B     [expr $E/(3.0*(1.0-2.0*$nu))]
nDMaterial PressureDependMultiYield $mTag 3 $mDen $G $B $fang 0.1 80 0.5 \
                                        $ptang 0.17 0.4 10  10 0.015 1.0

# Create element
set fBulk 2.2e6
set fDen  1.0
set alpha 1.5e-5
element SSPbrickUP 1 1 2 3 4 5 6 7 8 $mTag $fBulk $fDen $perm $perm $perm \
                                     $eNot $alpha

# Create recorders
recorder Node    -file disp.out   -time -nodeRange 5 8 -dof 3 disp
recorder Node    -file press.out  -time -nodeRange 1 8 -dof 4 vel
recorder Element -file stress.out -time stress
recorder Element -file strain.out -time strain

# Create analysis
constraints Penalty 1.0e17 1.0e17
test        NormDispIncr 1.0e-4 20 1
algorithm   Newton
numberer    RCM
system      BandGeneral
integrator  Newmark 0.5 0.25
rayleigh    $a0 0.0 $a1 0.0
analysis    Transient

# Apply confinement pressure
```

quad/brick elements have recorders to easily output the stress/strain at the integration points of the element

We use the Penalty constraints here because during the shearing phase, all of the pore pressure dof will be free.

Single Element Testing

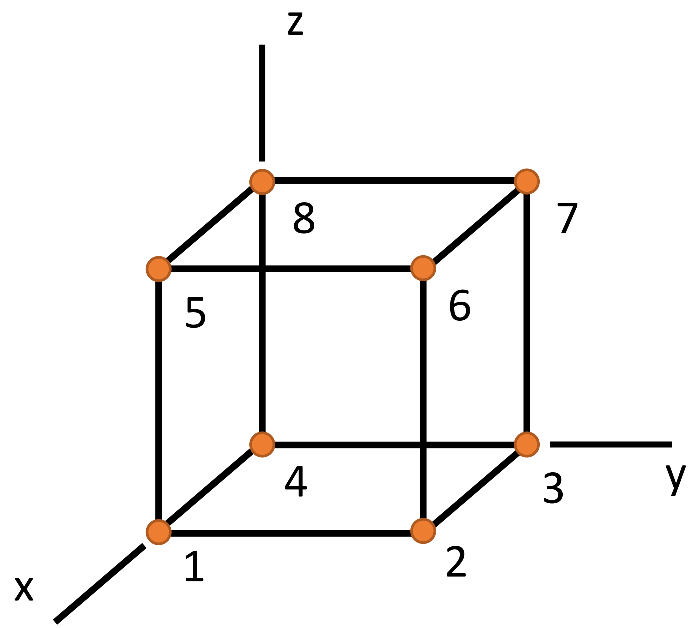
QuakeCoRE
NZ Centre for Earthquake Resilience

ctcTest.tcl (~/QuakeCoRE/OpenSe...dule3_geotechnicalExamples) - VIM

```
# Apply confinement pressure
set pNode [expr $pConf / 4.0]
pattern Plain 1 {Series -time {0 10000 1e10} -values {0 1 1} -factor 1} {
    load 1  $pNode  0.0    0.0    0.0
    load 2  $pNode  $pNode 0.0    0.0
    load 3  0.0     $pNode 0.0    0.0
    load 5  $pNode  0.0    $pNode 0.0
    load 6  $pNode  $pNode $pNode 0.0
    load 7  0.0     $pNode $pNode 0.0
    load 8  0.0     0.0    $pNode 0.0
}
analyze 100 100

# Let the model rest and waves damp out
analyze 10  1000

# Close drainage valves
for {set x 1} {$x<9} {incr x} {
   remove sp $x 4
}
analyze 5 0.1

updateMaterialStage -material 1 -stage 1

analyze 5 0.1

# Read vertical displacement of top plane
set vertDisp [nodeDisp 5 3]
# Apply deviatoric strain
set eDisp [expr 1+$devDisp/$vertDisp]
eval "timeSeries Path 5 -time {0 20001 20301 1e10
pattern Plain 2 5 {
    sp 5  3 $vertDisp
    sp 6  3 $vertDisp
    sp 7  3 $vertDisp
    sp 8  3 $vertDisp
}
```

The confinement pressure is applied in a Plain loadPattern with the imbedded timeSeries shown here.

Note that we divide the desired mean pressure by 4 when applying nodal forces.



Single Element Testing

QuakeCoRE
NZ Centre for Earthquake Resilience

ctcTest.tcl (~/QuakeCoRE/OpenSe...dule3_geotechnicalExamples) - VIM

```
# Apply confinement pressure
set pNode [expr $pConf / 4.0]
pattern Plain 1 {Series -time {0 10000 1e10} -values {0 1 1} -factor 1} {
    load 1  $pNode  0.0     0.0     0.0
    load 2  $pNode  $pNode  0.0     0.0
    load 3  0.0     $pNode  0.0     0.0
    load 5  $pNode  0.0     $pNode  0.0
    load 6  $pNode  $pNode  $pNode  0.0
    load 7  0.0     $pNode  $pNode  0.0
    load 8  0.0     0.0     $pNode  0.0
}
analyze 100 100

# Let the model rest and waves damp out
analyze 10  1000

# Close drainage valves
for {set x 1} {$x<9} {incr x} {
   remove sp $x 4
}
analyze 5 0.1

updateMaterialStage -material 1 -stage 1

analyze 5 0.1

# Read vertical displacement of top plane
set vertDisp [nodeDisp 5 3]
# Apply deviatoric strain
set eDisp [expr 1+$devDisp/$vertDisp]
eval "timeSeries Path 5 -time {0 20001 20301 1e10
pattern Plain 2 5 {
    sp 5  3 $vertDisp
    sp 6  3 $vertDisp
    sp 7  3 $vertDisp
    sp 8  3 $vertDisp
}
```

This a Transient analysis, so we must specify an analysis time step. Because we want the shearing phase to start from essentially static conditions, we use very large time steps to ensure that any waves created by the loading have damped out.

This technique only works because the material is elastic during this phase!

Single Element Testing

ctcTest.tcl (~/QuakeCoRE/OpenSe...dule3_geotechnicalExamples) - VIM

```
# Apply confinement pressure
set pNode [expr $pConf / 4.0]
pattern Plain 1 {Series -time {0 10000 1e10} -values {0 1 1} -factor 1} {
    load 1  $pNode  0.0     0.0     0.0
    load 2  $pNode  $pNode  0.0     0.0
    load 3  0.0     $pNode  0.0     0.0
    load 5  $pNode  0.0     $pNode  0.0
    load 6  $pNode  $pNode  $pNode  0.0
    load 7  0.0     $pNode  $pNode  0.0
    load 8  0.0     0.0     $pNode  0.0
}
analyze 100 100

# Let the model rest and waves damp out
analyze 10  1000

# Close drainage valves
for {set x 1} {$x<9} {incr x} {
   remove sp $x 4
}
analyze 5 0.1

updateMaterialStage -material 1 -stage 1

analyze 5 0.1

# Read vertical displacement of top plane
set vertDisp [nodeDisp 5 3]
# Apply deviatoric strain
set eDisp [expr 1+$devDisp/$vertDisp]
eval "timeSeries Path 5 -time {0 20001 20301 1e10
pattern Plain 2 5 {
    sp 5  3 $vertDisp
    sp 6  3 $vertDisp
    sp 7  3 $vertDisp
    sp 8  3 $vertDisp
}
```

After we are satisfied that hydrostatic stress conditions have been appropriately applied, we remove the constraints we specified on the pore pressure dof (4^{th} dof) of each node to "close the drainage valves". This is accomplished using the remove command

http://opensees.berkeley.edu/wiki/index.php/Remove_Command

Single Element Testing

ctcTest.tcl (~/QuakeCoRE/OpenSe...dule3_geotechnicalExamples) - VIM

```
# Apply confinement pressure
set pNode [expr $pConf / 4.0]
pattern Plain 1 {Series -time {0 10000 1e10} -values {0 1 1} -factor 1} {
    load 1  $pNode  0.0     0.0     0.0
    load 2  $pNode  $pNode  0.0     0.0
    load 3  0.0     $pNode  0.0     0.0
    load 5  $pNode  0.0     $pNode  0.0
    load 6  $pNode  $pNode  $pNode  0.0
    load 7  0.0     $pNode  $pNode  0.0
    load 8  0.0     0.0     $pNode  0.0
}
analyze 100 100

# Let the model rest and waves damp out
analyze 10  1000

# Close drainage valves
for {set x 1} {$x<9} {incr x} {
   remove sp $x 4
}
analyze 5 0.1

updateMaterialStage -material 1 -stage 1

analyze 5 0.1

# Read vertical displacement of top plane
set vertDisp [nodeDisp 5 3]
# Apply deviatoric strain
set eDisp [expr 1+$devDisp/$vertDisp]
eval "timeSeries Path 5 -time {0 20001 20301 1e10
pattern Plain 2 5 {
    sp 5  3 $vertDisp
    sp 6  3 $vertDisp
    sp 7  3 $vertDisp
    sp 8  3 $vertDisp
}
```

The next step is to instruct the nDMaterial object to consider elastoplastic response using the updateMaterialStage command. The default state for the PressureDependMultiYield model is linear elastic behaviour.

http://opensees.berkeley.edu/wiki/index.php/UpdateMaterialStage

Single Element Testing

QuakeCoRE
NZ Centre for Earthquake Resilience

ctcTest.tcl (~/QuakeCoRE/OpenSe...dule3_

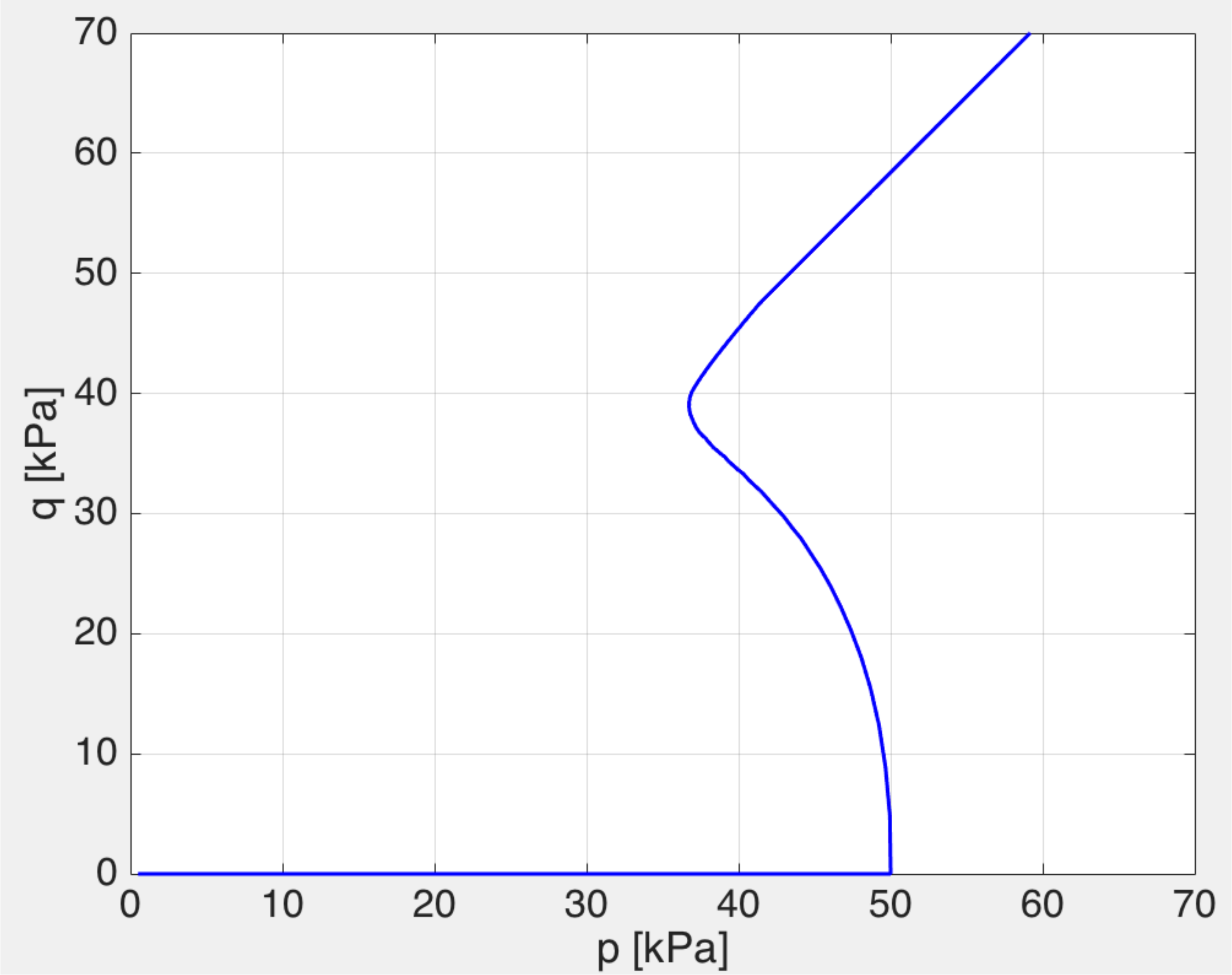
```
    load 6  $pNode  $pNode $pNode 0.0
    load 7  0.0     $pNode $pNode 0.0
    load 8  0.0     0.0    $pNode 0.0
}
analyze 100 100

# Let the model rest and waves damp out
analyze 10  1000

# Close drainage valves
for {set x 1} {$x<9} {incr x} {
   remove sp $x 4
}
analyze 5 0.1

updateMaterialStage -material 1 -stage 1

analyze 5 0.1

# Read vertical displacement of top plane
set vertDisp [nodeDisp 5 3]
# Apply deviatoric strain
set eDisp [expr 1+$devDisp/$vertDisp]
eval "timeSeries Path 5 -time {0 20001 20301 1e10} -values {0 1 $eDisp $eDisp}"
pattern Plain 2 5 {
    sp 5  3 $vertDisp
    sp 6  3 $vertDisp
    sp 7  3 $vertDisp
    sp 8  3 $vertDisp
}

# Set number and length of time steps
set dT      0.1
set numStep 3000

analyze $numStep $dT

wipe
```

The shearing phase is the last part of the analysis. The deviator stress is applied under strain control in this example by specifying the displacements of the upper nodes in the loadPattern using the sp command

http://opensees.berkeley.edu/wiki/index.php/Sp_Command

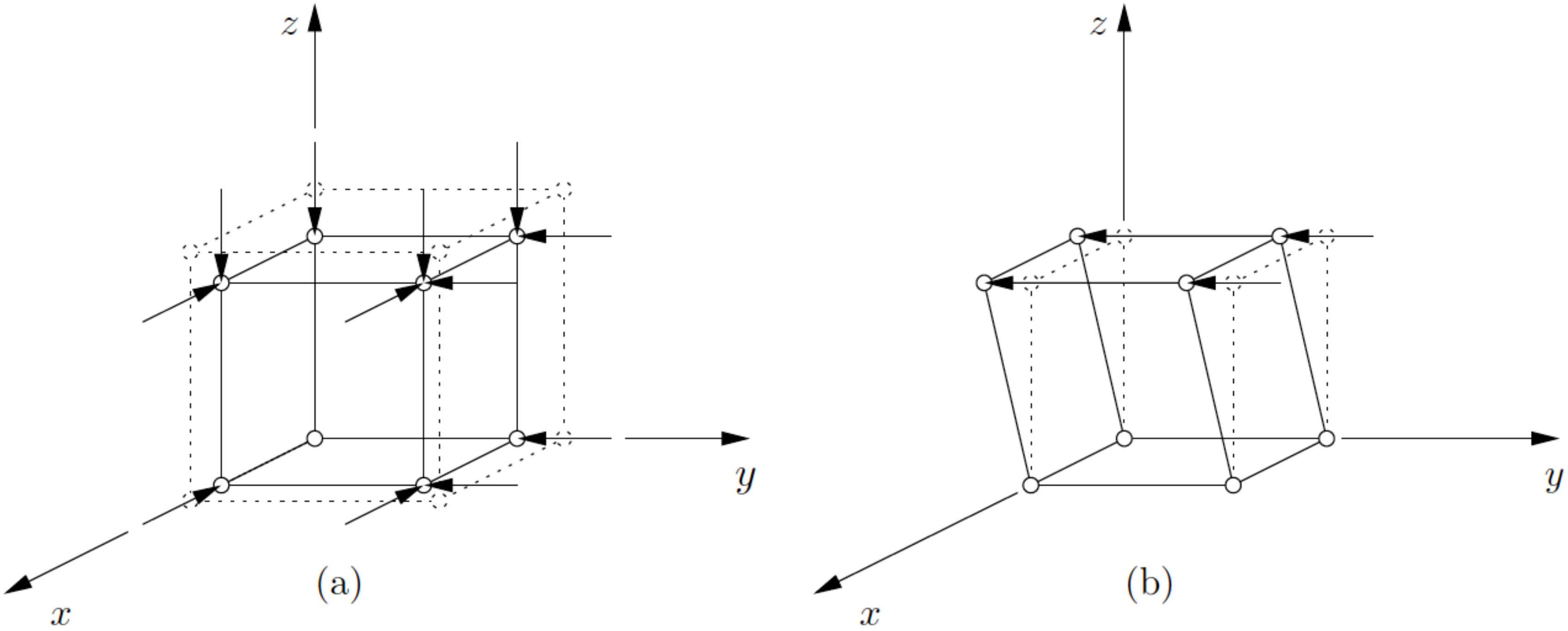
Single-Element Testing

From the recorded stresses, we can create a plot of the stress path for this single element simulated CTC test as shown below.



Single-Element Testing

We can simulate other laboratory tests using single element models and a similar approach to what we just saw.



For example, a direct shear test can be modeled using a single element as shown in this schematic. The model file for a cyclic direct shear test has been made available through this workshop.

Total Stress Site Response Analysis

QuakeCoRE
NZ Centre for Earthquake Resilience

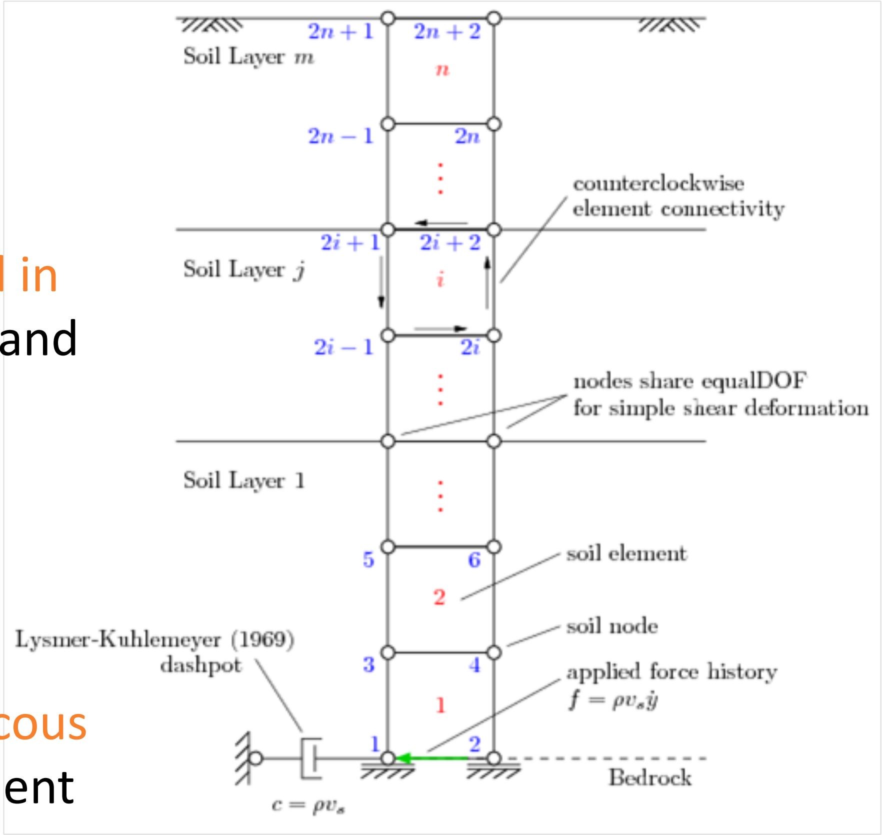
This example describes how to run a total stress site response analysis in OpenSees.

A layered or homogenous soil profile is modeled in 2D using nodes with 2 DOF.

Periodic boundary conditions are enforced in horiz. direction using the equalDOF command

Soil constitutive models include:

PressureDependMultiYield
PressureIndependMultiYield



A compliant base is considered using a viscous dashpot modeled using a zeroLength element and the viscous uniaxialMaterial .

A detailed discussion is available on the OpenSees practical examples page at:
http://opensees.berkeley.edu/wiki/index.php/Site_Response_Analysis_of_a_Layered_Soil_Column_(Total_Stress_Analysis)

Total Stress Site Response Analysis

QuakeCoRE
NZ Centre for Earthquake Resilience

freeFieldSingle.tcl (~/QuakeCoRE/OpenSeesWor...Files/module3_geotechnicalExamples/site) - VIM2

```
#-----------------------------------------------------------------------------------------
#  1. DEFINE ANALYSIS PARAMETERS
#-----------------------------------------------------------------------------------------

#---SOIL GEOMETRY
# thicknesses of soil profile (m)
set soilThick  40.0

#---MATERIAL PROPERTIES
# soil mass density (Mg/m^3)
set rho         1.7
# soil shear wave velocity (m/s)
set Vs          250.0
# soil shear modulus (kPa)
set G           [expr $rho*$Vs*$Vs]
# poisson's ratio of soil
set nu          0.0
# soil elastic modulus (kPa)
set E           [expr 2*$G*(1+$nu)]
# soil bulk modulus (kPa)
set bulk        [expr $E/(3*(1-2*$nu))]
# soil cohesion (kPa)
set cohesion    95.0
# peak shear strain
set gammaPeak   0.05
# soil friction angle
set phi         0.0
# reference pressure
set refPress    80.0
# pressure dependency coefficient
set pressCoeff  0.0
# bedrock shear wave velocity (m/s)
set rockVS      760
# bedrock mass density (Mg/m^3)
set rockDen     2.4

#---GROUND MOTION PARAMETERS
# time step in ground motion record
```

The material and geometric parameters for the soil column are placed in the beginning of the file to make things easier to find and change.

This example considers a single 40 m thick layer of soil.

A model file for a layered soil profile follows the exact same steps, it is just more complicated, so we are using a simpler case.

Total Stress Site Response Analysis

QuakeCoRE
NZ Centre for Earthquake Resilience

freeFieldSingle.tcl (~/QuakeCoRE/OpenSee.../module3_geotechnicalExamples/site) - VIM2

```
# bedrock shear wave velocity (m/s)
set rockVS       760
# bedrock mass density (Mg/m^3)
set rockDen      2.4

#---GROUND MOTION PARAMETERS
# time step in ground motion record
set motionDT         0.005
# number of steps in ground motion record
set motionSteps      7990

#---WAVELENGTH PARAMETERS
# highest frequency desired to be well resolved (Hz)
set fMax        100.0
# wavelength of highest resolved frequency
set wave        [expr $Vs/$fMax]
# number of elements per one wavelength
set nEle        10

#---RAYLEIGH DAMPING PARAMETERS
set pi      3.141592654
# damping ratio
set damp    0.02
# lower frequency
set omega1      [expr 2*$pi*0.2]
# upper frequency
set omega2      [expr 2*$pi*20]
# damping coefficients
set a0      [expr 2*$damp*$omega1*$omega2/($omega1 + $omega2)]
set a1      [expr 2*$damp/($omega1 + $omega2)]
puts "damping coefficients: a_0 = $a0;  a_1 = $a1"

#---ANALYSIS PARAMETERS
# Newmark parameters
set gamma            0.5
set beta             0.25

#---------------------------------------------------------------------------------------------
```

We also define some analysis parameters here, including the ground motion details, the Rayleigh damping terms, and Newmark integrator terms.

We also specify some parameters that will be used to determine the size of the elements in the model. Here we have specified 100 Hz as the highest frequency we want well resolved (10 elements in one wavelength). The element size will be determined based on this information.

Total Stress Site Response Analysis

QuakeCoRE
NZ Centre for Earthquake Resilience

freeFieldSingle.tcl (~/QuakeCoRE/OpenSee.../module3_geotechnicalExamples/site) - VIM2

```
#-----------------------------------------------------------------------
#  2. DEFINE MESH GEOMETRY
#-----------------------------------------------------------------------

# trial vertical element size
set hTrial    [expr $wave/$nEle]

# trial number of elements
set nTrial    [expr $soilThick/$hTrial]
# round up if not an integer number of elements
if { $nTrial > [expr floor($soilThick/$hTrial)] } {
    set numEleY  [expr int(floor($soilThick/$hTrial)+1)]
} else {
    set numEleY  [expr int($nTrial)]
}
puts "number of verticl elements: $numEleY"

# final vertical size of elements (m)
set sizeEleY  [expr $soilThick/$numEleY]
puts "vertical size of elements: $sizeEleY"

# define horizontal size of elements (m)
set sizeEleX  $sizeEleY
puts "horizontal size of elements: $sizeEleX"

# number of nodes in vertical direction
set numNodeY  [expr 2*($numEleY + 1)]

#-----------------------------------------------------------------------------------------
#  3. DEFINE NODES FOR SOIL ELEMENTS
#-----------------------------------------------------------------------------------------

# soil nodes are created in 2 dimensions, with 2 translational dof
model BasicBuilder -ndm 2 -ndf 2

set yCoord     0.0
set count      0
# loop over nodes
```

The element size definition is here. We set a trial number of elements based on the wavelength computed previously, then make sure an integer number of elements will be in the model (can't have 2.34516 elems).

The number of elems defines the vertical element size and then the number of nodes.

Total Stress Site Response Analysis

QuakeCoRE
NZ Centre for Earthquake Resilience

freeFieldSingle.tcl (~/QuakeCoRE/OpenSee.../module3_geotechnicalExamples/site) - VIM2

```
#-----------------------------------------------------------------------------------------
#  3. DEFINE NODES FOR SOIL ELEMENTS
#-----------------------------------------------------------------------------------------

# soil nodes are created in 2 dimensions, with 2 translational dof
model BasicBuilder -ndm 2 -ndf 2

set yCoord     0.0
set count      0
# loop over nodes
for {set j 1} {$j <= $numNodeY} {incr j 2} {
    node    $j            0.0          [expr $yCoord + $count*$sizeEleY]
    node    [expr $j+1]   $sizeEleX    [expr $yCoord + $count*$sizeEleY]

    set count [expr $count+1]
}
puts "Finished creating all soil nodes..."

#----------------------------------------------------------
#  4. DEFINE DASHPOT NODES
#----------------------------------------------------------

node 2000 0.0 0.0
node 2001 0.0 0.0

puts "Finished creating dashpot nodes..."

#----------------------------------------------------------
#  5. DEFINE BOUNDARY CONDITIONS AND EQUAL DOF
#----------------------------------------------------------

# define fixity of base nodes
fix 1 0 1
fix 2 0 1

# define fixity of dashpot nodes
fix  2000  1 1
fix  2001  0 1
```

The nodes for the soil column are created in a loop.

Taking advantage of tcl scripting like this saves a ton of time (and space in your model file).

For example, in this case there are 160 elements in the model, which corresponds to 322 nodes. Entering those by hand would not be very fun.

Total Stress Site Response Analysis

QuakeCoRE
NZ Centre for Earthquake Resilience

freeFieldSingle.tcl (~/QuakeCoRE/OpenSe

```
puts "Finished creating all soil nodes..."

#-----------------------------------------------
#  4. DEFINE DASHPOT NODES
#-----------------------------------------------

node 2000 0.0 0.0
node 2001 0.0 0.0

puts "Finished creating dashpot nodes..."

#-----------------------------------------------
#  5. DEFINE BOUNDARY CONDITIONS AND EQUAL DOF
#-----------------------------------------------

# define fixity of base nodes
fix 1 0 1
fix 2 0 1

# define fixity of dashpot nodes
fix  2000  1 1
fix  2001  0 1

# define equal DOF for simple shear deformation of soil elements
for {set k 3} {$k <= $numNodeY} {incr k 2} {
    equalDOF  $k  [expr $k+1]  1 2
}

# define equal DOF for dashpot and base soil nodes
equalDOF 1    2 1
equalDOF 1 2001 1

puts "Finished creating all boundary conditions and equalDOF..."

#---------------------------------------------------------------------------------------------
#  6. DEFINE SOIL MATERIALS
#---------------------------------------------------------------------------------------------
```

The compliance of the underlying material is considered in the model using a dashpot (after Lysmer) and the ground motion application technique of Joyner and Chen (1975).

For this modeling approach, a viscous dashpot is placed at the base of the soil column.

Here we also implement the periodic boundary conditions using the equalDOF command.

Total Stress Site Response Analysis

QuakeCoRE
NZ Centre for Earthquake Resilience

freeFieldSingle.tcl (~/QuakeCoRE/OpenSee.../module3_geotechnicalExamples/site) - VIM2

```
#------------------------------------------------------------------------------------------
#  6. DEFINE SOIL MATERIALS
#------------------------------------------------------------------------------------------

nDMaterial PressureIndependMultiYield 1 2 $rho $G $bulk $cohesion $gammaPeak \
                                          $phi $refPress $pressCoeff 25

puts "Finished creating all soil materials..."

#-----------------------------------------------------
#  7. DEFINE SOIL ELEMENTS
#-----------------------------------------------------

set wgtX  0.0
set wgtY  [expr -9.81*$rho]

# loop over elements
for {set j 1} {$j <= $numEleY} {incr j 1} {

        set nI  [expr 2*$j - 1]
        set nJ  [expr $nI + 1]
        set nK  [expr $nI + 3]
        set nL  [expr $nI + 2]

        element SSPquad $j $nI $nJ $nK $nL 1 "PlaneStrain" 1.0 $wgtX $wgtY
}

puts "Finished creating all soil elements..."

#------------------------------------------------------------------------------------------
#  8. DEFINE MATERIAL AND ELEMENTS FOR VISCOUS DAMPERS
#------------------------------------------------------------------------------------------

# dashpot coefficient
set mC     [expr $sizeEleX*$rockDen*$rockVS]

# material
uniaxialMaterial Viscous 4000 $mC 1
```

The nDMaterial and elements are defined here.

The elements are defined in a loop as well. Setting this up initially takes a bit of thought, but it definitely pays off.

Total Stress Site Response Analysis

QuakeCoRE
NZ Centre for Earthquake Resilience

freeFieldSingle.tcl (~/QuakeCoRE/OpenSee.../mod

```
#  8. DEFINE MATERIAL AND ELEMENTS FOR VISCOUS DAMPERS
#-----------------------------------------------------------

# dashpot coefficient
set mC     [expr $sizeEleX*$rockDen*$rockVS]

# material
uniaxialMaterial Viscous 4000 $mC 1

# elements
element zeroLength 5000 2000 2001 -mat 4000 -dir 1

puts "Finished creating dashpot material and element..."

#-----------------------------------------------------------
#  8. CREATE GRAVITY RECORDERS
#-----------------------------------------------------------

# record nodal displacments, velocities, and accelerations at each time step
recorder Node -file Gdisplacement.out -time  -nodeRange 1 $numNodeY -dof 1 2  disp
recorder Node -file Gvelocity.out     -time  -nodeRange 1 $numNodeY -dof 1 2  vel
recorder Node -file Gacceleration.out -time  -nodeRange 1 $numNodeY -dof 1 2  accel

# record stress and strain at each gauss point in the soil elements
recorder Element -file Gstress.out   -time  -eleRange  1   $numEleY   stress
recorder Element -file Gstrain.out   -time  -eleRange  1   $numEleY   strain

puts "Finished creating gravity recorders..."

#------------------------------------------------------------------------------------------
#  9. APPLY GRAVITY LOADING
#------------------------------------------------------------------------------------------

constraints Transformation
test        NormDispIncr 1e-5 30 1
algorithm   Newton
numberer    RCM
system      ProfileSPD
```

The viscous uniaxialMaterial is used with a zeroLength element to define the dashpot at the base of the soil column.

Recorders for the gravity stage of the analysis are also defined here.

Total Stress Site Response Analysis

QuakeCoRE
NZ Centre for Earthquake Resilience

freeFieldSingle.tcl (~/QuakeCoRE/OpenSee.../module3_geotechnicalExamples/site) - VIM2

```
#  8. CREATE GRAVITY RECORDERS
#------------------------------------------------------------------------------------------

# record nodal displacments, velocities, and accelerations at each time step
recorder Node -file Gdisplacement.out -time  -nodeRange 1 $numNodeY -dof 1 2  disp
recorder Node -file Gvelocity.out     -time  -nodeRange 1 $numNodeY -dof 1 2  vel
recorder Node -file Gacceleration.out -time  -nodeRange 1 $numNodeY -dof 1 2  accel

# record stress and strain at each gauss point in the soil elements
recorder Element -file Gstress.out   -time  -eleRange  1   $numEleY   stress
recorder Element -file Gstrain.out   -time  -eleRange  1   $numEleY   strain

puts "Finished creating gravity recorders..."

#-------------------------------------------------------
#  9. APPLY GRAVITY LOADING
#-------------------------------------------------------

constraints Transformation
test        NormDispIncr 1e-5 30 1
algorithm   Newton
numberer    RCM
system      ProfileSPD
integrator  Newmark $gamma $beta
analysis    Transient

analyze     10 5.0e2

puts "Finished with elastic gravity analysis..."

# update material to consider elastoplastic behavior
updateMaterialStage -material 1 -stage 1

# plastic gravity loading
analyze     40 5.0e2

puts "Finished with plastic gravity analysis..."
```

The gravity analysis phase (that ensures there is a proper initial state of stress in the soil before ground motion application) is defined here.

We apply 10 steps with elastic response in the material, update the material stage to consider plastic response, then apply 40 more steps.

Large time steps are used to damp out the waves generated by loading.

Total Stress Site Response Analysis

freeFieldSingle.tcl (~/QuakeCoRE/OpenSee.../module3_geotechnicalExamples/site) - VIM2

```
#------------------------------------------------------------------------------------------
#  10. CREATE POST-GRAVITY RECORDERS
#------------------------------------------------------------------------------------------

# reset time and analysis
setTime 0.0
wipeAnalysis

# record nodal displacments, velocities, and accelerations at each time step
recorder Node -file displacement.out -time -dT $motionDT -nodeRange 1 $numNodeY -dof 1 2  disp
recorder Node -file velocity.out     -time -dT $motionDT -nodeRange 1 $numNodeY -dof 1 2  vel
recorder Node -file acceleration.out -time -dT $motionDT -nodeRange 1 $numNodeY -dof 1 2  accel

# record stress and strain in the soil elements
recorder Element -file stress.out   -time -dT $motionDT -eleRange  1   $numEleY   stress
recorder Element -file strain.out   -time -dT $motionDT -eleRange  1   $numEleY   strain

puts "Finished creating all recorders..."

#-----------------------------------------------------
#  11. DYNAMIC ANALYSIS
#-----------------------------------------------------

# define constant factor for applied velocity
set cFactor [expr $sizeEleX*$rockDen*$rockVS]

# define velocity time history file
set velocityFile velocityHistory.out

# timeseries object for applied force history
set mSeries "Path -dt $motionDT -filePath $velocityFile

# loading object
pattern Plain 10 $mSeries {
    load 1 1.0 0.0
}
puts "Dynamic loading created..."
```

The setTime command is used to reset the analysis time back to 0.0, the loads from the gravity phase remain in place, this just starts the clock back to the start for the shaking phase.

The wipeAnalysis command removes all of the previous analysis objects from the model. We will define a new set for the next stage.

Total Stress Site Response Analysis

QuakeCoRE
NZ Centre for Earthquake Resilience

freeFieldSingle.tcl (~/QuakeCoRE/OpenSee.../module3_geotechnicalExamples/site) - VIM2

```
#------------------------------------------------------------------------------------------
#  11. DYNAMIC ANALYSIS
#------------------------------------------------------------------------------------------

# define constant factor for applied velocity
set cFactor [expr $sizeEleX*$rockDen*$rockVS]

# define velocity time history file
set velocityFile velocityHistory.out

# timeseries object for applied force history
set mSeries "Path -dt $motionDT -filePath $velocityFile -factor $cFactor"

# loading object
pattern Plain 10 $mSeries {
    load 1 1.0 0.0
}
puts "Dynamic loading created..."

#---DETERMINE STABLE ANALYSIS TIME STEP USING CFL CONDITION
# duration of ground motion (s)
set duration  [expr $motionDT*$motionSteps]
# trial analysis time step
set kTrial    [expr $sizeEleX/(pow($Vs,0.5))]
# define time step and number of steps for analysis
if { $motionDT <= $kTrial } {
    set nSteps  $motionSteps
    set dT      $motionDT
} else {
    set nSteps  [expr floor($duration/$kTrial)+1]
    set dT      [expr $duration/$nSteps]
}
puts "number of steps in analysis: $nSteps"
puts "analysis time step: $dT"

# analysis objects
constraints Transformation
```

After the method of Joyner and Chen (1975), the ground motion is applied to the base of the soil column as a force time history

This force is proportional to the product of the density and shear wave velocity of the underlying material (assumed to be rock) and the velocity time history of the ground motion record.

A Plain loadPattern is used for this purpose.

Total Stress Site Response Analysis

QuakeCoRE
NZ Centre for Earthquake Resilience

freeFieldSingle.tcl (~/QuakeCoRE/OpenSee.../module3_geotechnicalExamples/site) - VIM2

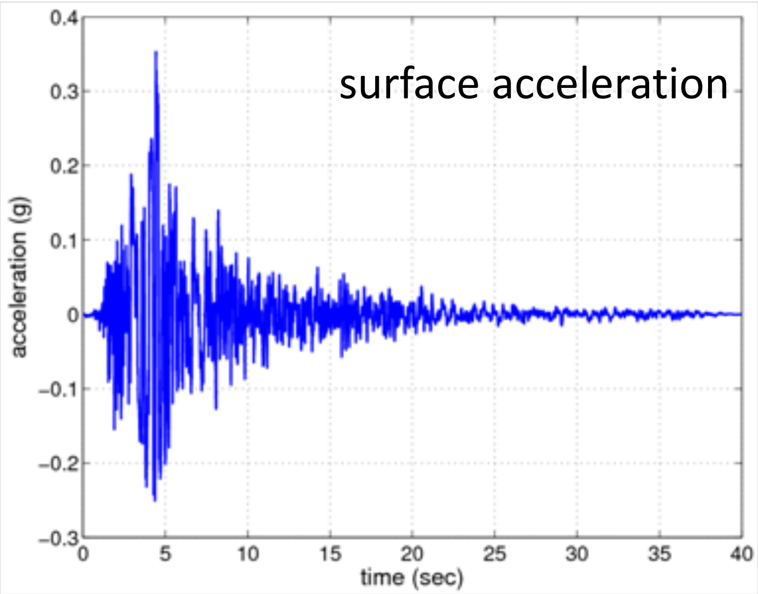
```
set mSeries "Path -dt $motionDT -filePath $velocityFile -factor $cFactor"

# loading object
pattern Plain 10 $mSeries {
    load 1 1.0 0.0
}
puts "Dynamic loading created..."

#---DETERMINE STABLE ANALYSIS TIME STEP USING CFL CONDITION
# duration of ground motion (s)
set duration  [expr $motionDT*$motionSteps]
# trial analysis time step
set kTrial    [expr $sizeEleX/(pow($Vs,0.5))]
# define time step and number of steps for analysis
if { $motionDT <= $kTrial } {
    set nSteps  $motionSteps
    set dT      $motionDT
} else {
    set nSteps  [expr floor($duration/$kTrial)+1]
    set dT      [expr $duration/$nSteps]
}
puts "number of steps in analysis: $nSteps"
puts "analysis time step: $dT"

# analysis objects
constraints Transformation
test        NormDispIncr 1e-3 15 1
algorithm   Newton
numberer    RCM
system      ProfileSPD
integrator  Newmark $gamma $beta
rayleigh    $a0 $a1 0.0 0.0
analysis    Transient

analyze     $nSteps  $dT

puts "Finished with dynamic analysis..."
```

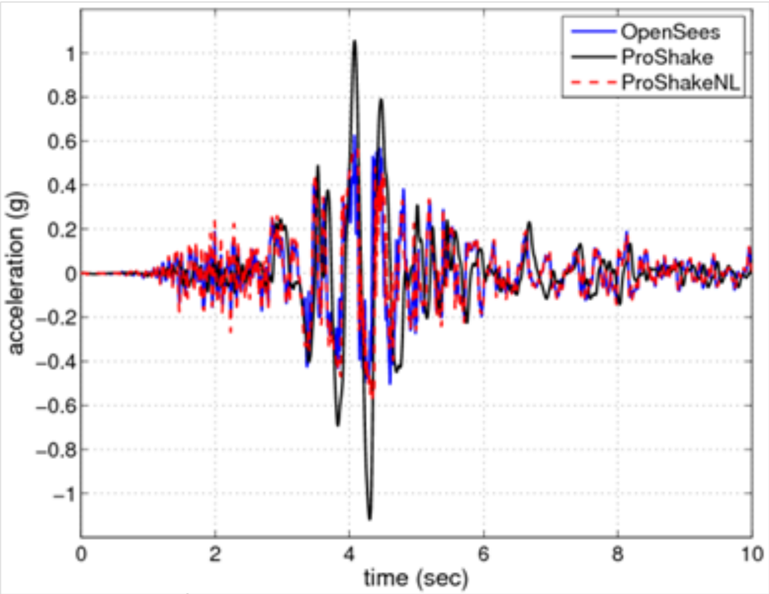
Before conducting the analysis using the analysis objects shown here, we use the Courant–Friedrichs–Lewy (CFL) condition to find a stable analysis time step based on the size and stiffness of the elements.

The time step of the ground motion will be used unless it is found that a smaller time step is needed for stability.

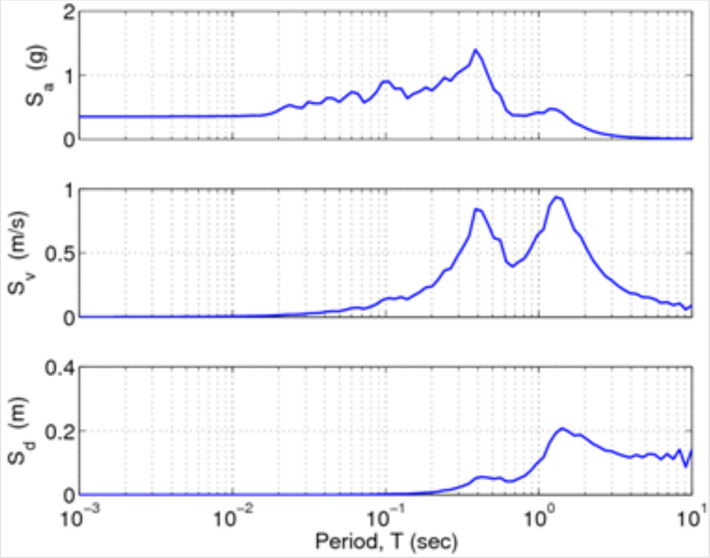
Total Stress Site Response Analysis

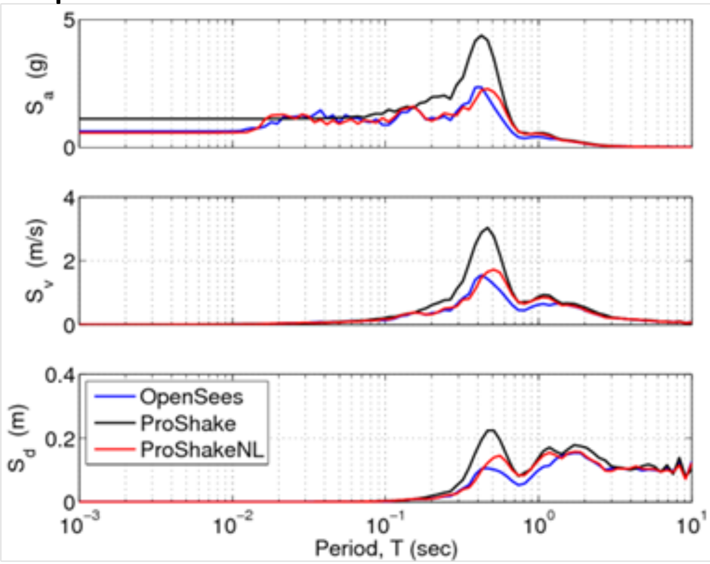
QuakeCoRE
NZ Centre for Earthquake Resilience





surface response spectra



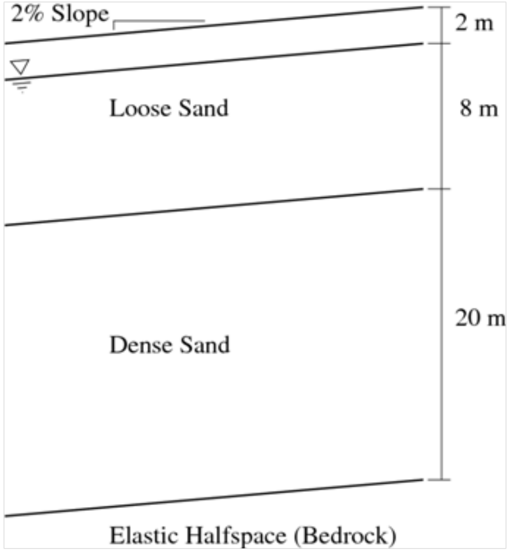


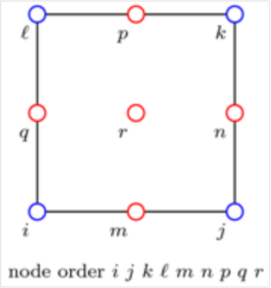
Effective Stress Site Response

The approach is similar to the total stress analysis. A layered soil profile is modeled in 2D with periodic displacement boundary conditions enforced using the equalDOF command and a compliant base is considered using a viscous dashpot modeled using a zeroLength element and the viscous uniaxialMaterial.

The 9_4_QuadUP element is used to model the soil. This element considers the interaction between the pore fluid and the solid soil skeleton, allowing for phenomena such as liquefaction to be modeled.

The PressureDependMultiYield02 constitutive model is used for the soil.

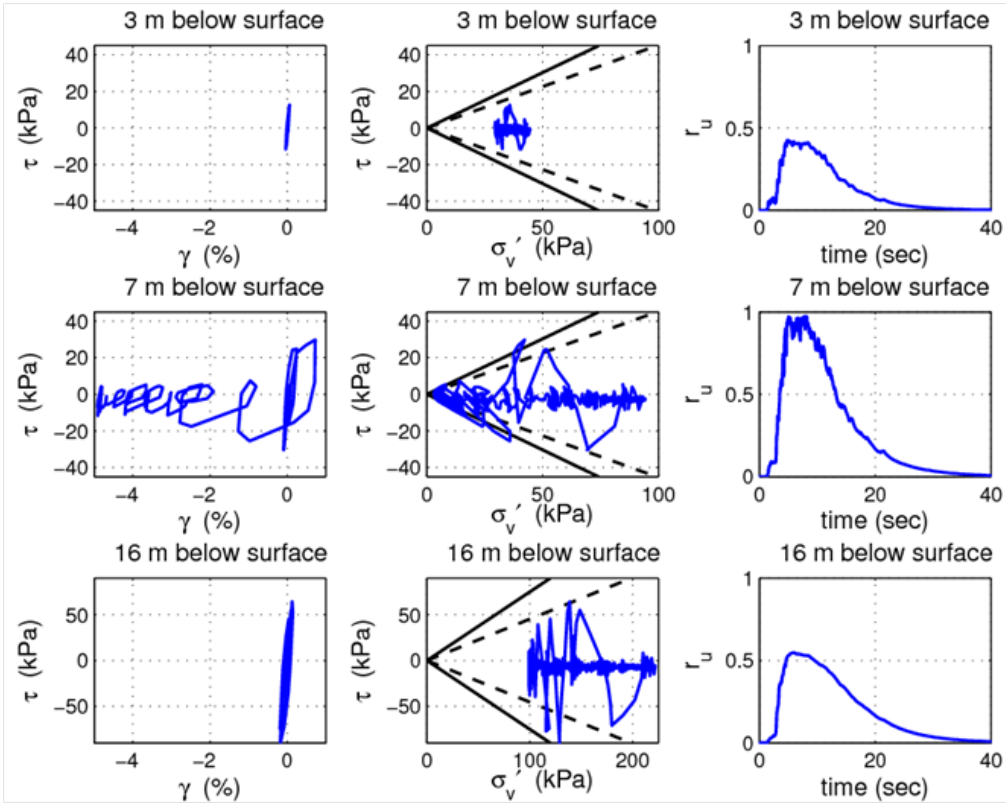




Site Response Analysis

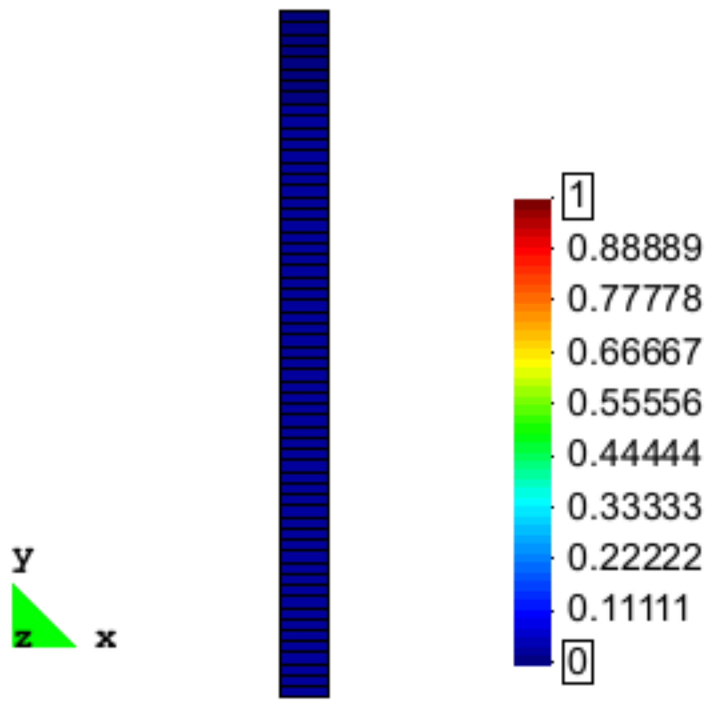
QuakeCoRE
NZ Centre for Earthquake Resilience

summary of soil behavior at three depths within the soil profile



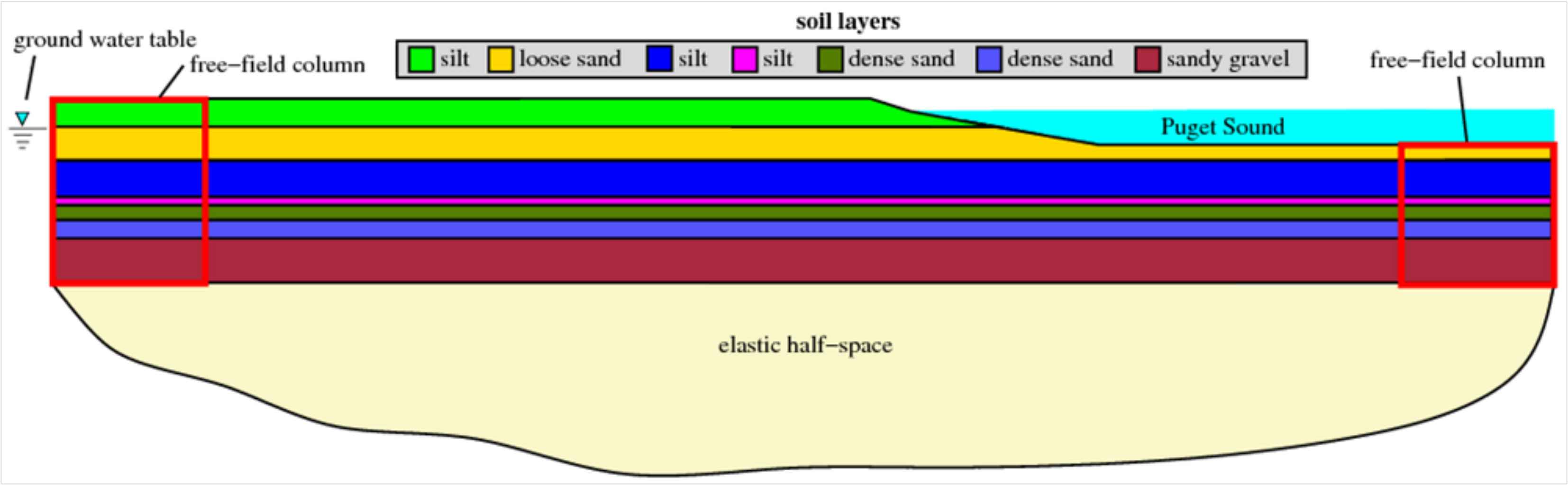
Site Response Analysis

displacement of soil column during analysis with contours of excess pore pressure ratio



Effective Stress Slope Analysis

Model problem:



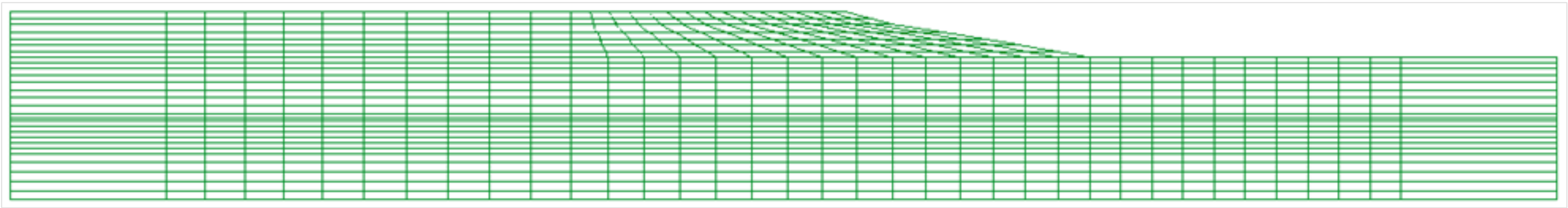
This example presents a 2D effective stress analysis of a slope subject to an earthquake ground motion.

The elements and constitutive models match those used in the site response analysis examples.

The free-field soil response is applied to the model using free-field columns which are much more massive than the adjacent soil.

Effective Stress Slope Analysis

Finite element mesh:



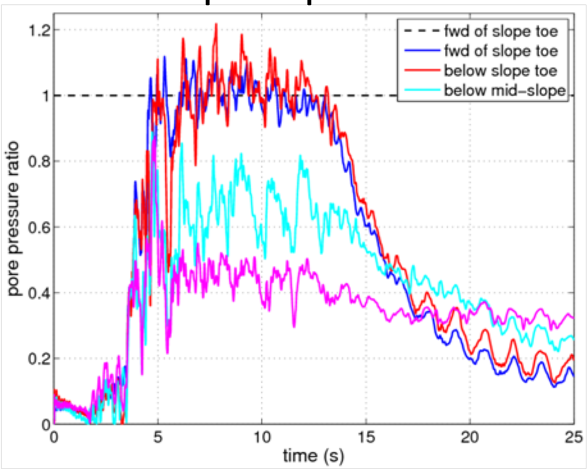
This example presents a 2D effective stress analysis of a slope subject to an earthquake ground motion.

The elements and constitutive models match those used in the site response analysis examples.

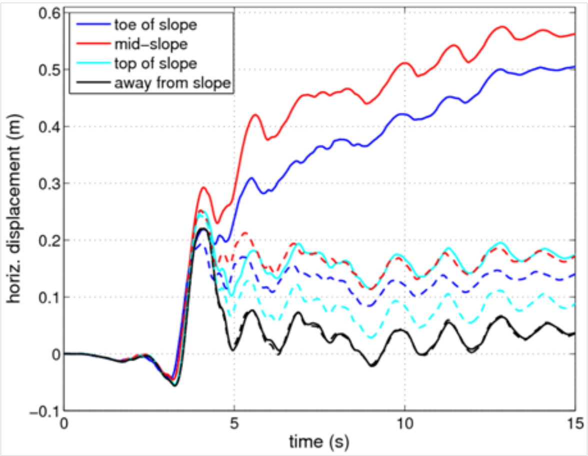
The free-field soil response is applied to the model using free-field columns which are much more massive than the adjacent soil.

Effective Stress Slope Analysis

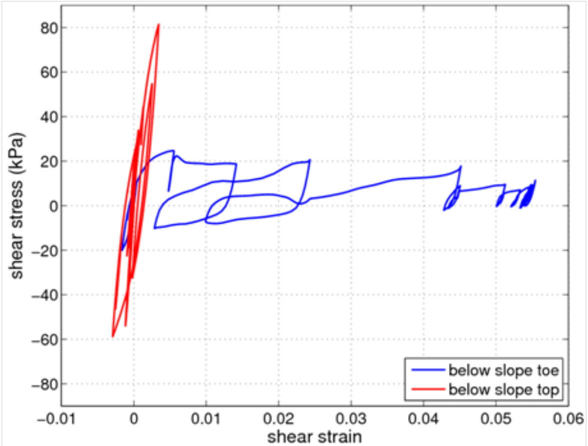
excess pore pressure ratio



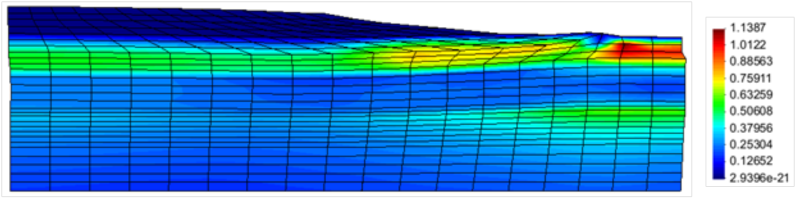
lateral displacement



shear stress-strain



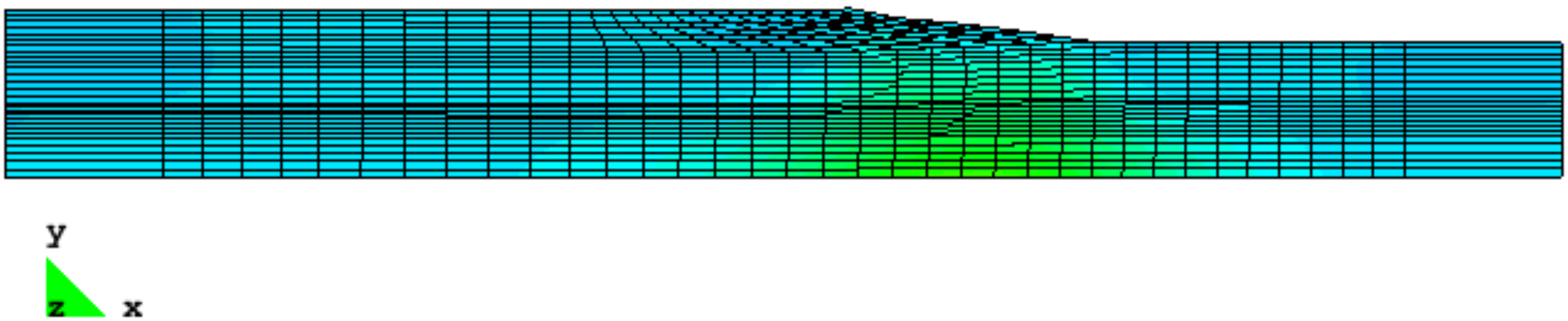
excess pore pressure ratio contours near slope



Effective Stress Slope Analysis

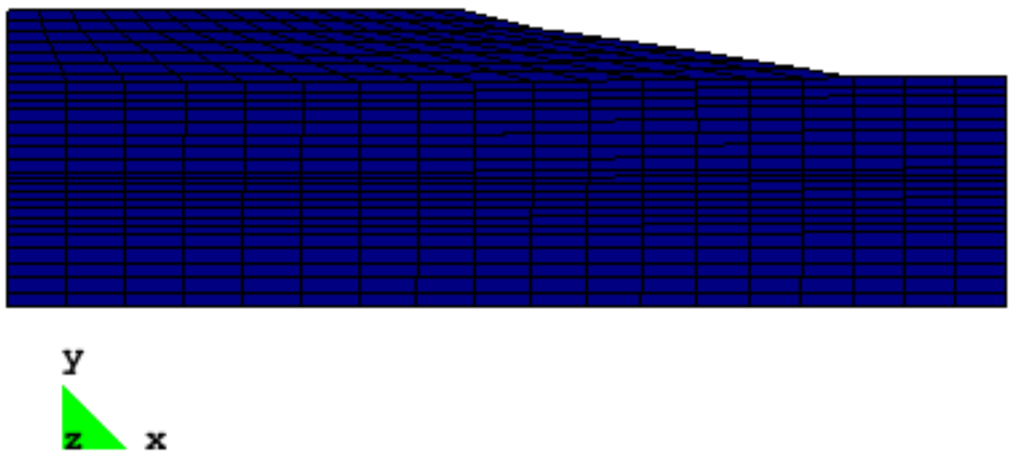
QuakeCoRE
NZ Centre for Earthquake Resilience

displacement of full mesh during analysis showing propagation of shear stress waves



Effective Stress Slope Analysis

displacement near the slope with contours of excess pore pressure ratio (red is r_u = 1.0)

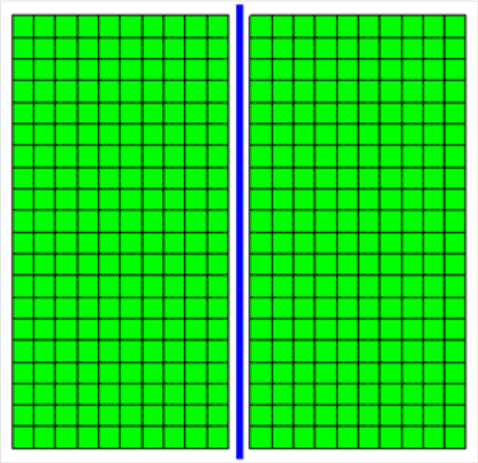


Progressive Excavation Analysis

This example presents a simulated excavation supported by a sheet pile wall using OpenSees.

The sheet pile wall is modeled using the dispBeamColumn element with an elastic fiber section for linear elastic constitutive behavior.

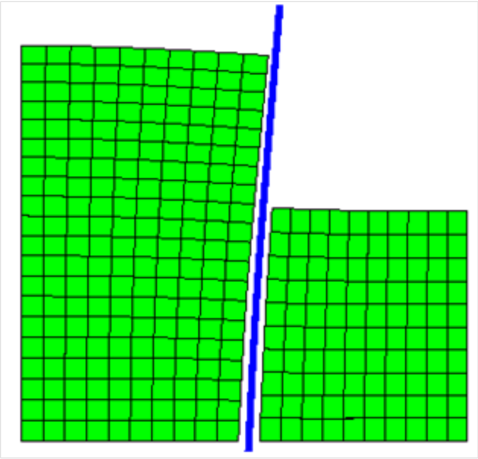
The soil-wall interface is modeled using the BeamContact2D element.



The InitialStateAnalysis feature is used to create the gravitational state of stress in the model without accompanying displacements.

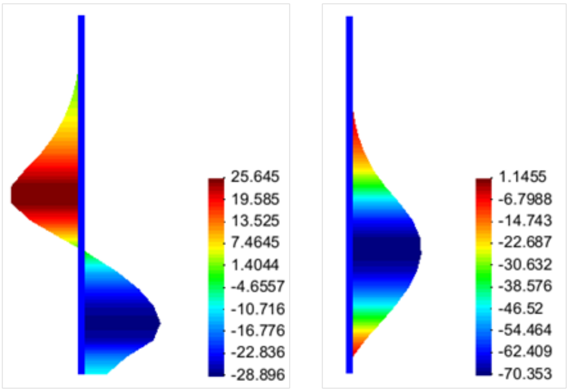
The plane strain formulation of the quad element is used for the soil with the PressureDependMultiYield nDMaterial for constitutive behavior.

Soil elements to the right of the wall are progressively removed to simulate an excavation.

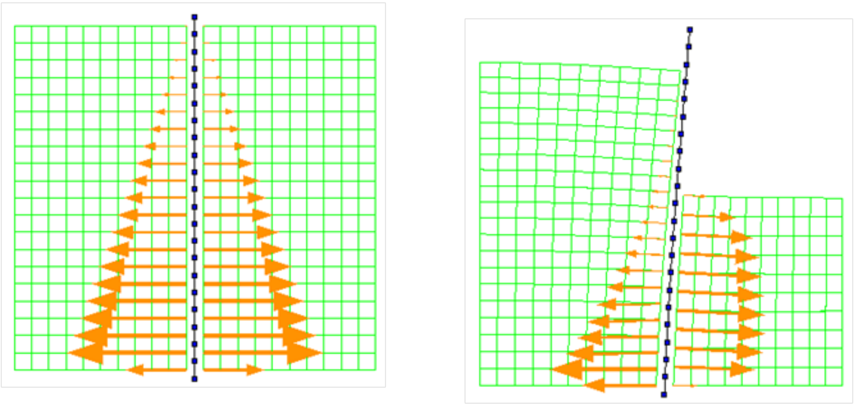


Progressive Excavation Analysis

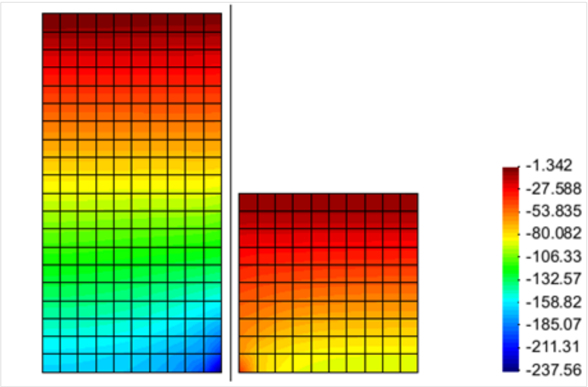
shear and moment in the wall



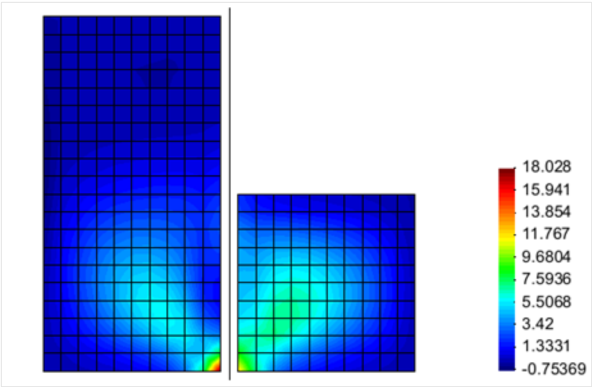
wall-soil contact forces



vertical stress contours

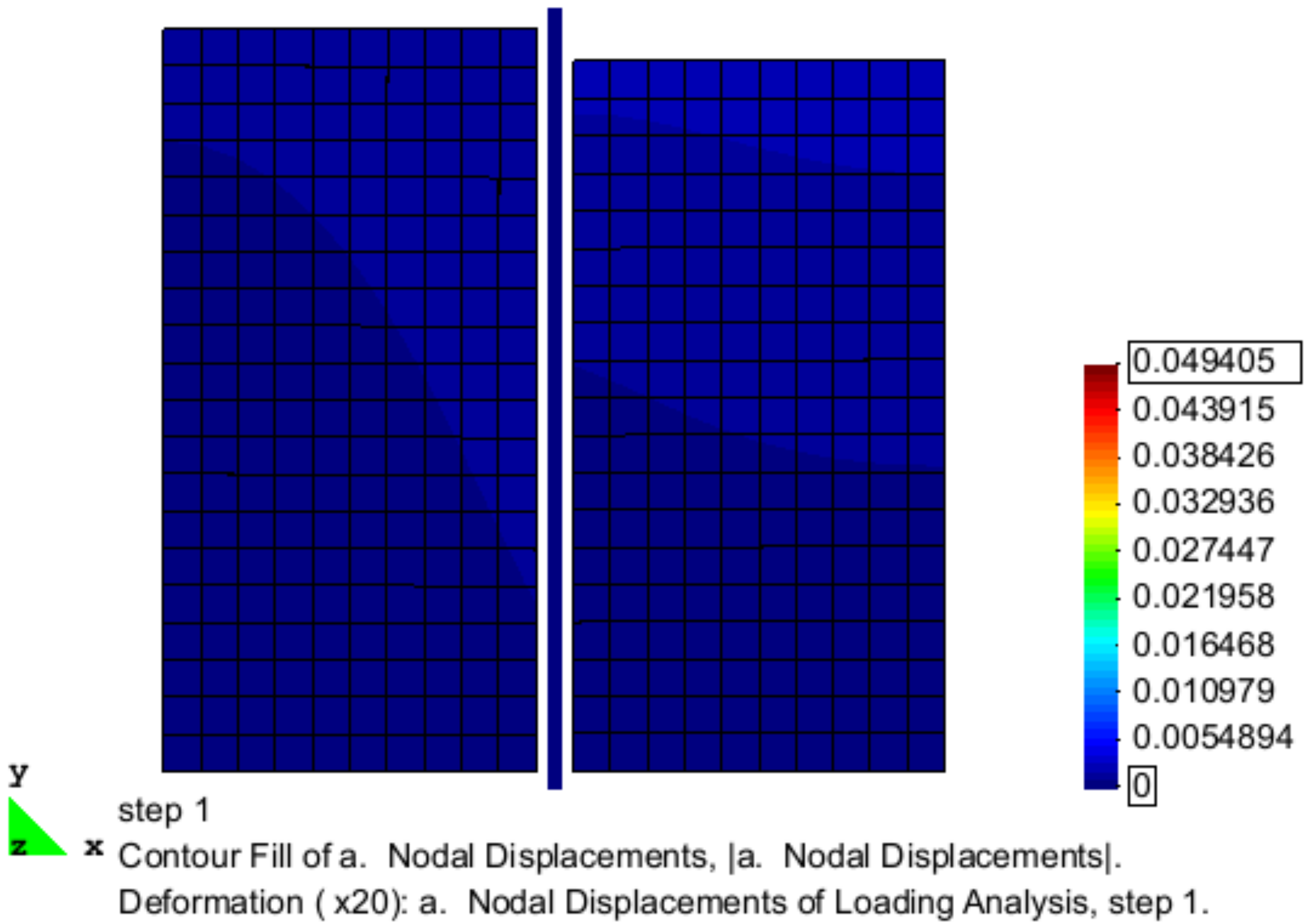


shear stress contours



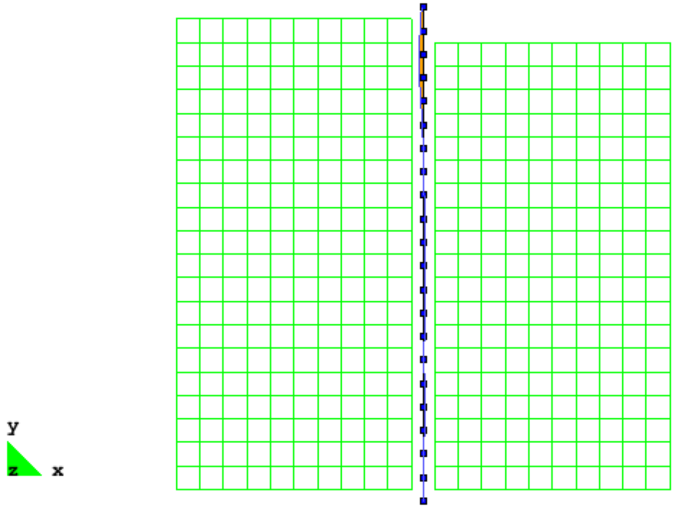
Progressive Excavation Analysis

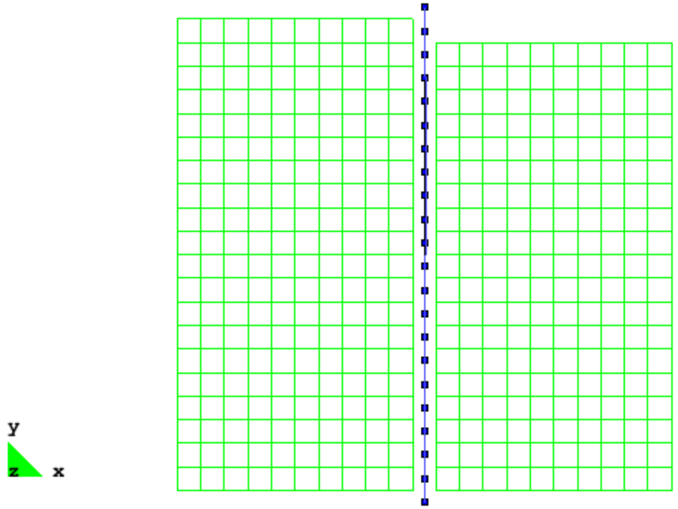
Nodal displacement magnitude during the excavation analysis



Progressive Excavation Analysis

shear and moment in the wall during the excavation analysis





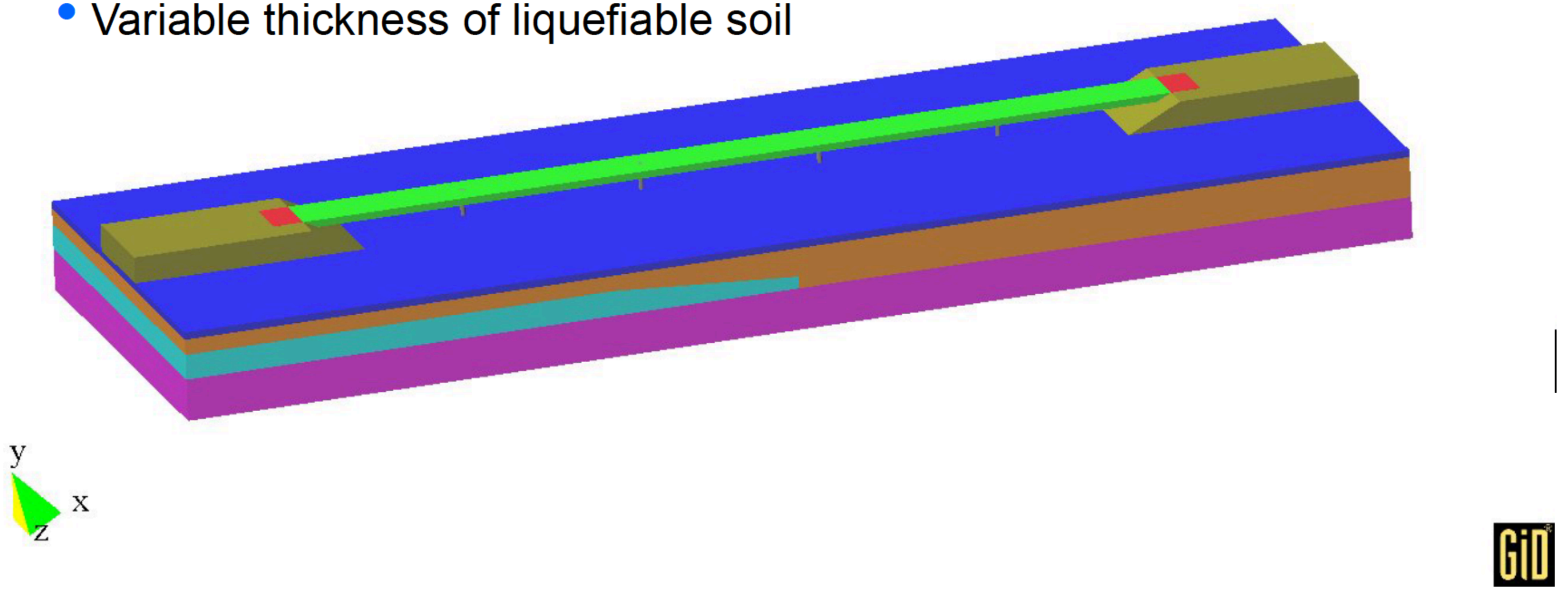
SFSI of Complete Bridge System

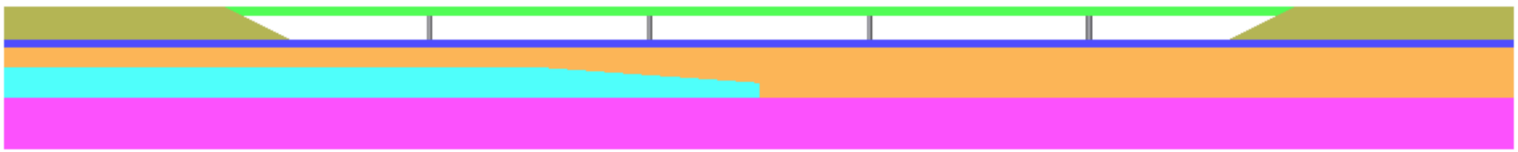
- Five-span bridge
- pile group foundation
- abutment
- liquefiable soil / various layers

- lateral spreading
- earthquake intensities
- uncertainties

SFSI of Complete Bridge System

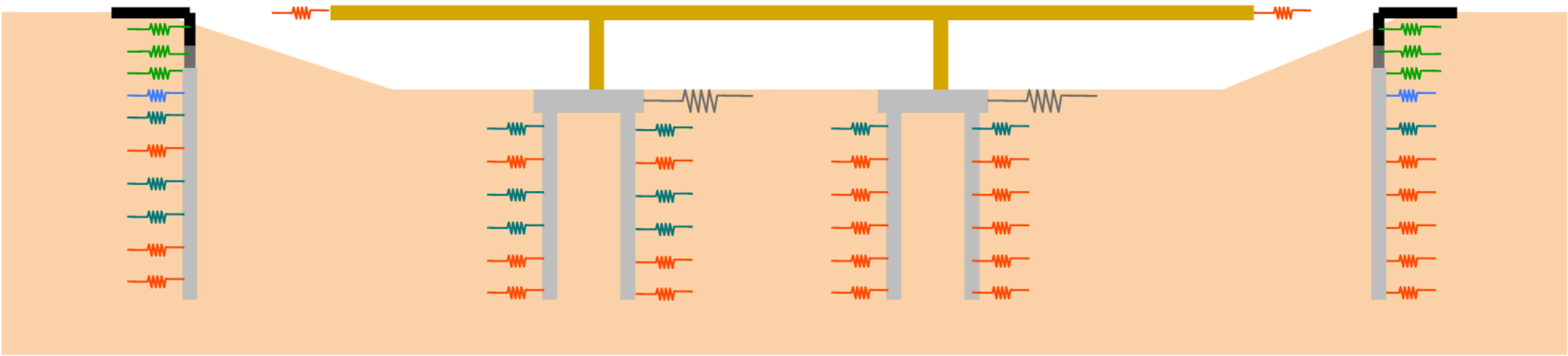
- Five-span bridge
- Approach embankments
- Variable thickness of liquefiable soil



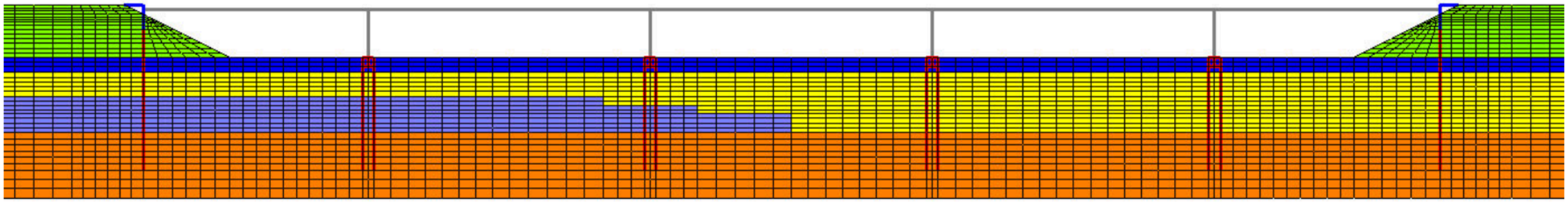


SFSI of Complete Bridge System

Bridge Idealization

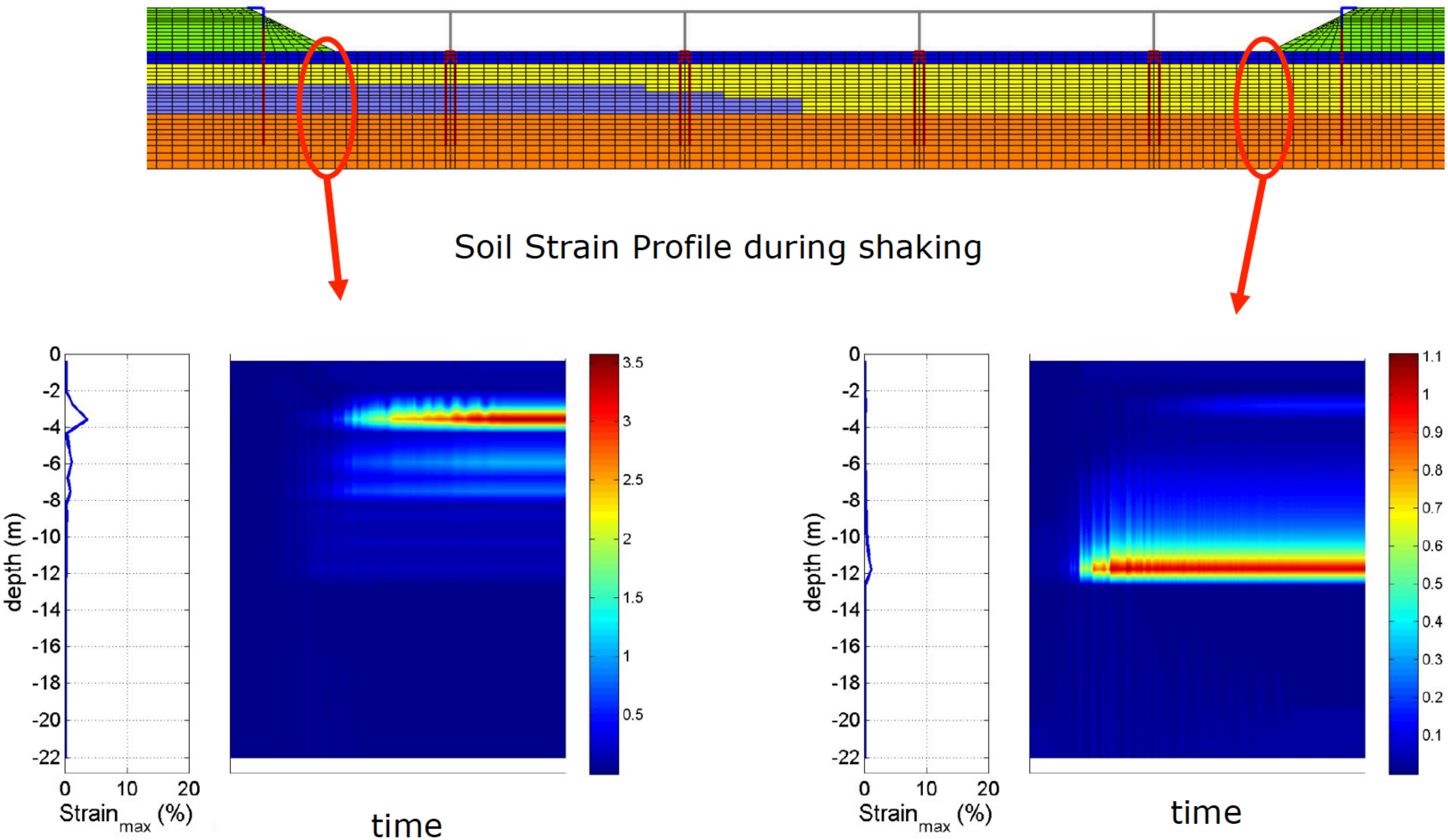


OpenSees model



SFSI of Complete Bridge System

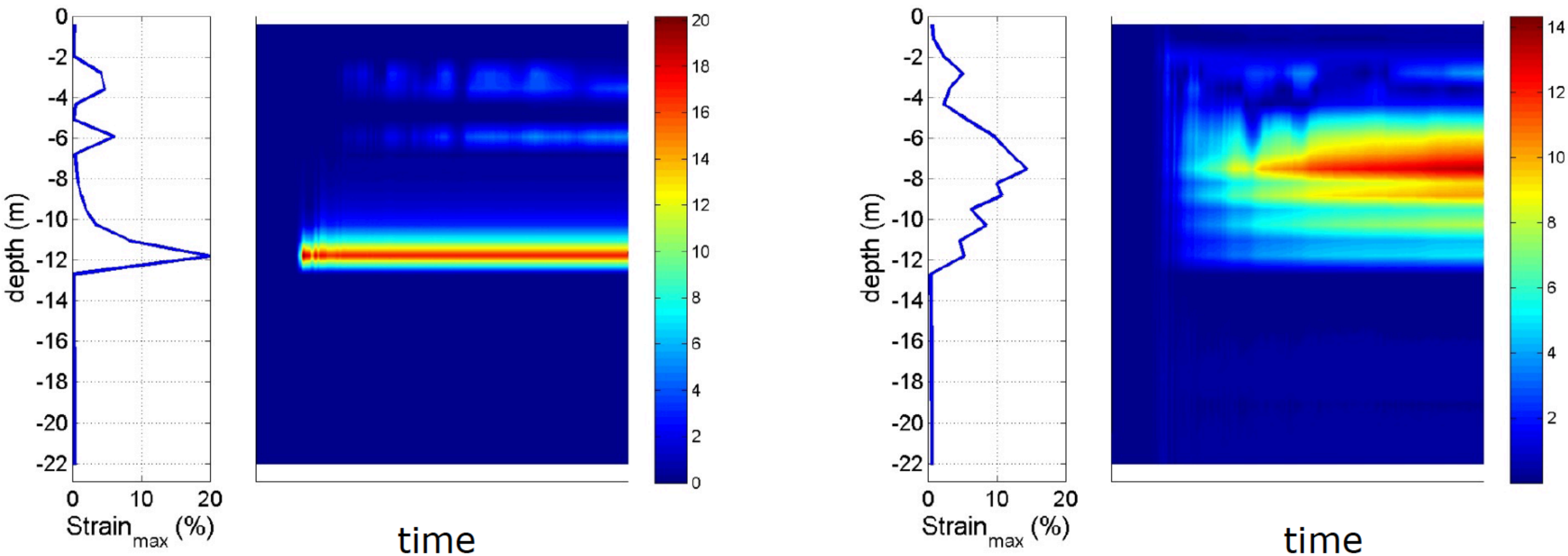
QuakeCoRE
NZ Centre for Earthquake Resilience



Northridge motion (0.25g)

SFSI of Complete Bridge System

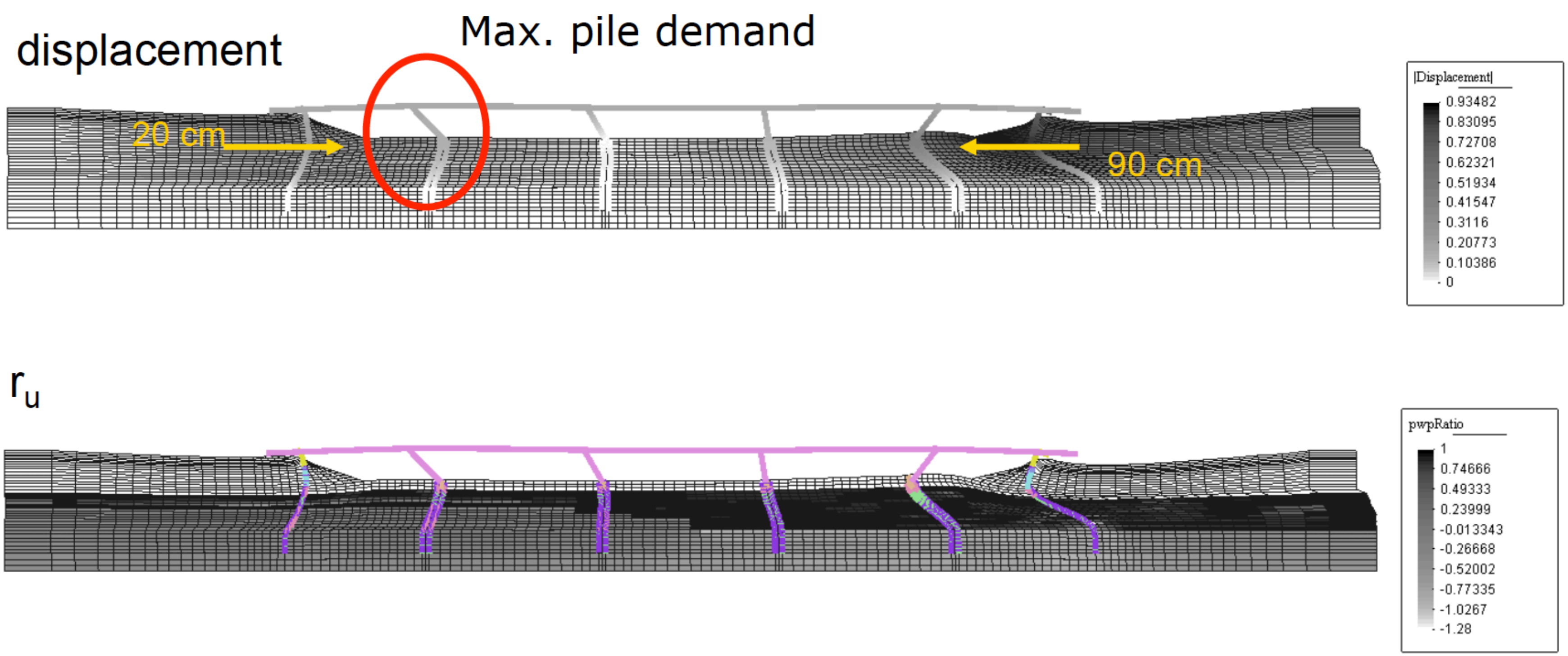
Soil Strain Profile during shaking



Loma Prieta (1.19g)

SFSI of Complete Bridge System

QuakeCoRE
NZ Centre for Earthquake Resilience



Erzincan, Turkey 1992 (a_{max} = 0.70g)

Thank you!

www.quakecore.nz