THE HONG KONG INSTITUTION OF ENGINEERS
香港工程師學會

Geotechnical Division
岩土分部

HKIE Geotechnical Division Seminar:
Common Pitfalls and Important Points to Note in Using Geotechnical Computer Programs (Geotechnical Computer Program Users Group Meeting No. 2)

Improving Modelling Accuracy

A Perspective from the Global Finite Element Scheme for Soil-Structure Interaction Problems

Dr. LI, Chao

Lambeth Associates Ltd.

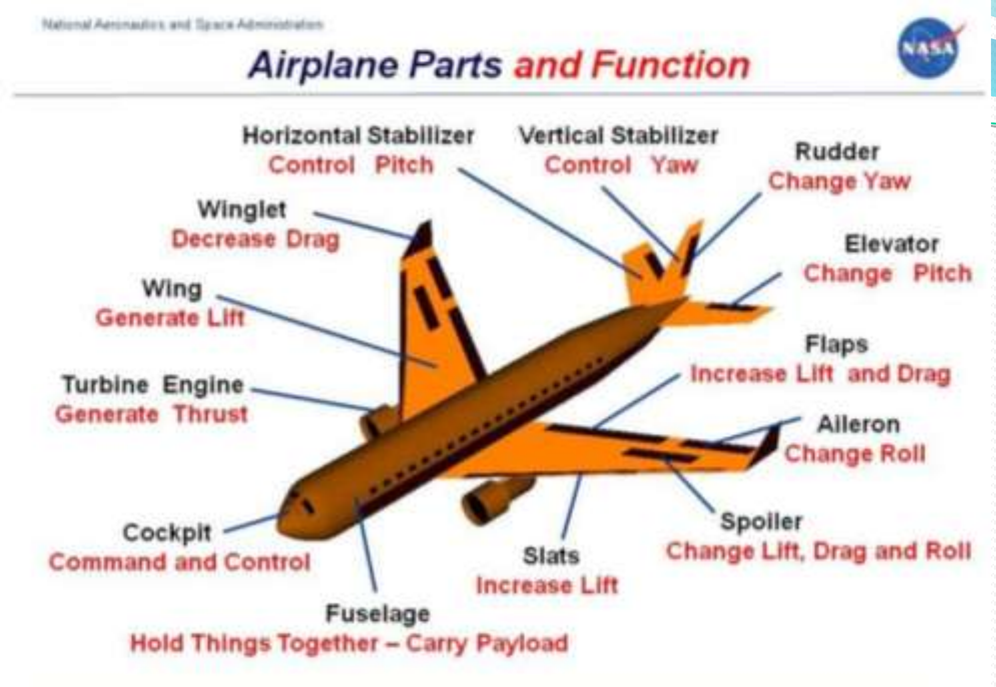
17-September-2014

Email: chao.li@lambeth.com.hk

Structure of Presentation

- Part 1: A glimpse into the FEM
  - Closer look to one element
- Part 2: How does it work ?
  - Input, output and solution process
- Part 3: Why does it work?
  - Brief introduction to the theory
- Part 4: What else can go wrong?
  - Nonlinearity
  - Convergence
  - Stability
  - Accuracy
- Part 5: Back to analysts' world

Part 1: A Glimpse into the Finite Element Method



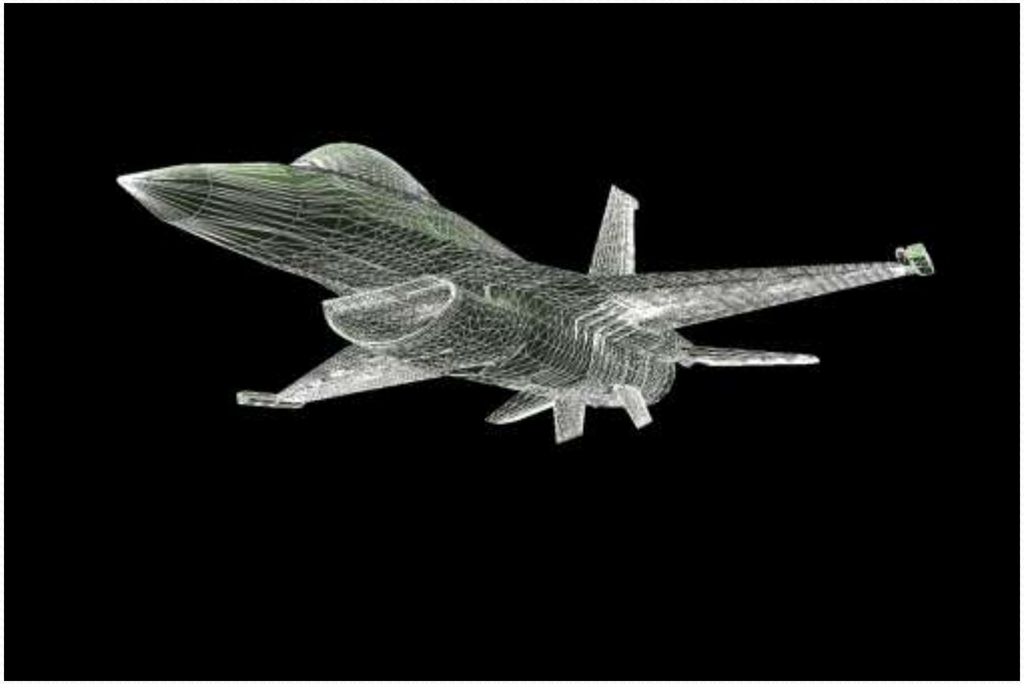


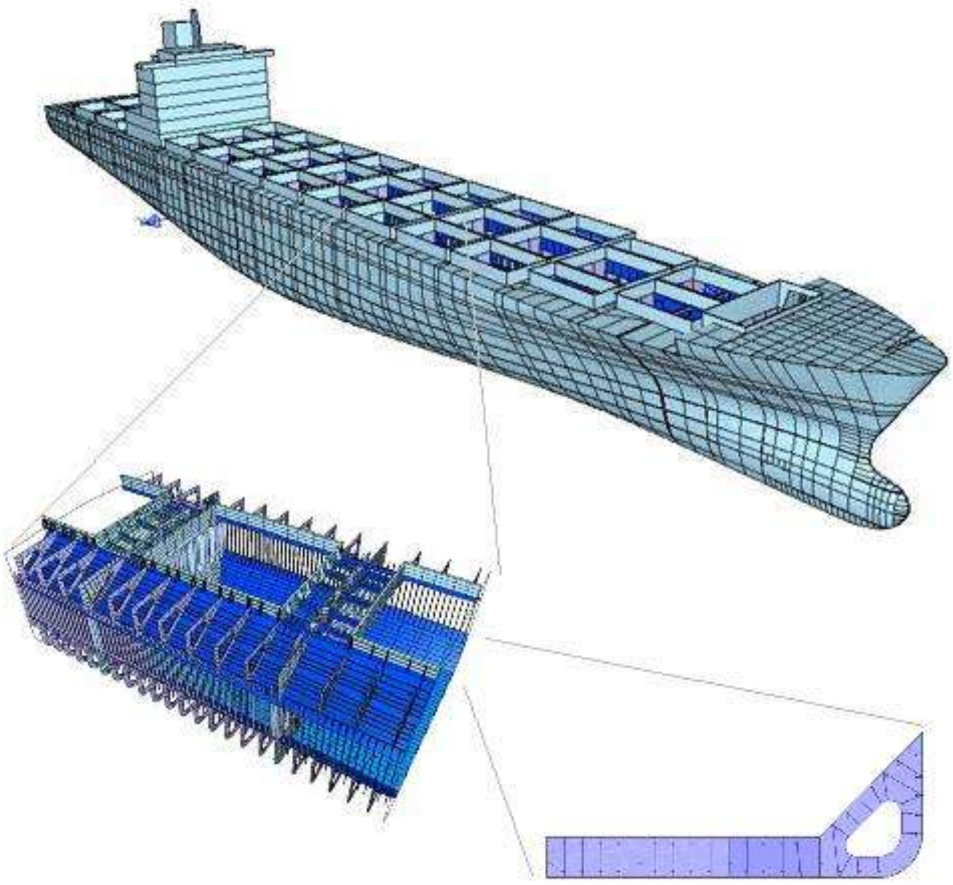
Illustration 1

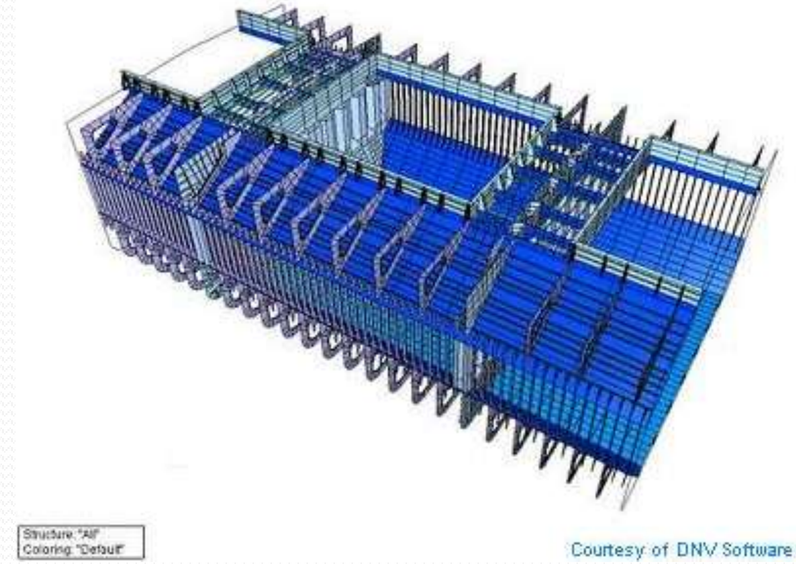
An airplane

A finite element model of an airplane

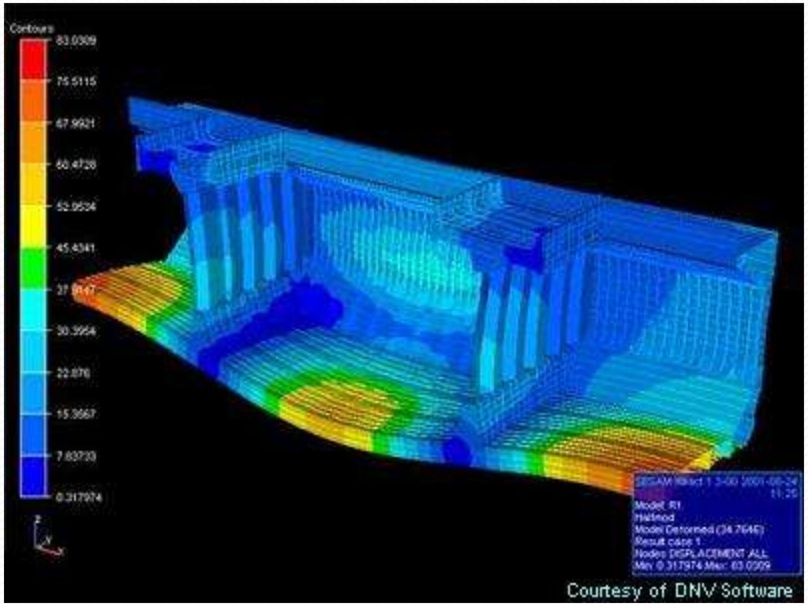
Are they equivalent?

Illustration 2: Hierarchy in FEM Simulation





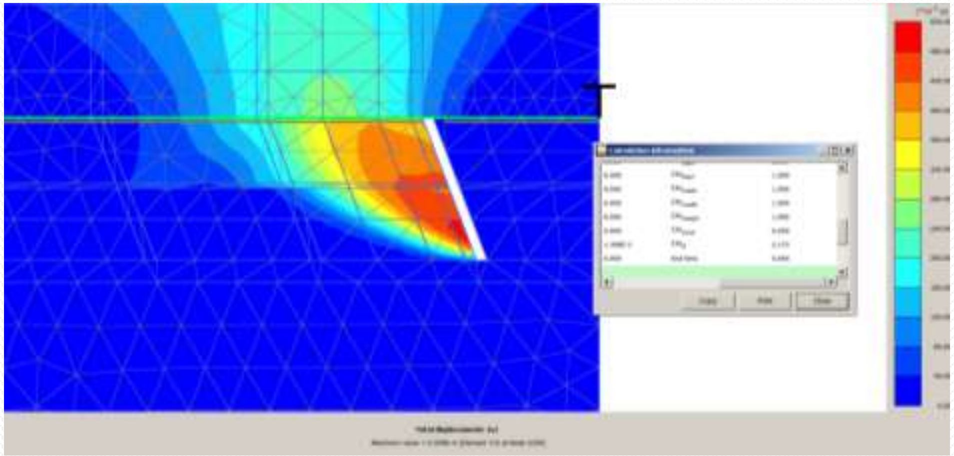
Courtesy of DNV Software



What is FEM?

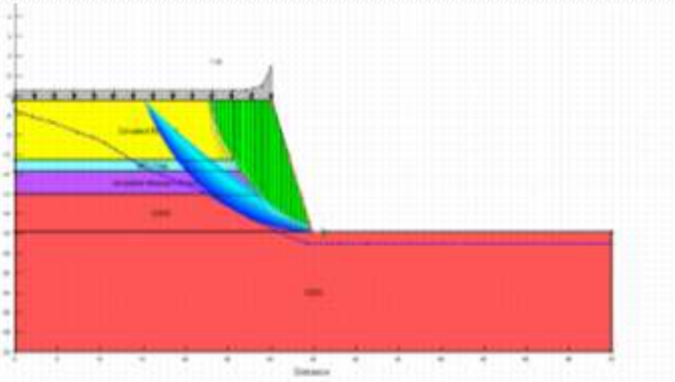
- FEM is
  - An approximate way of solving partial differential equations
  - A tool of computational mechanics
  - A fundamentally "more correct" method (compared with **limit equilibrium method**)
  - A stress-deformation method
  - An "accurate" method (**if and only if** accurate input information is given)
- Other parallel methods include
  - Boundary Element Method
  - Finite Difference Method
  - Finite Volume Method

Finite Element Method and Limit Equilibrium Method for FOS analysis



FOS = 2.173!

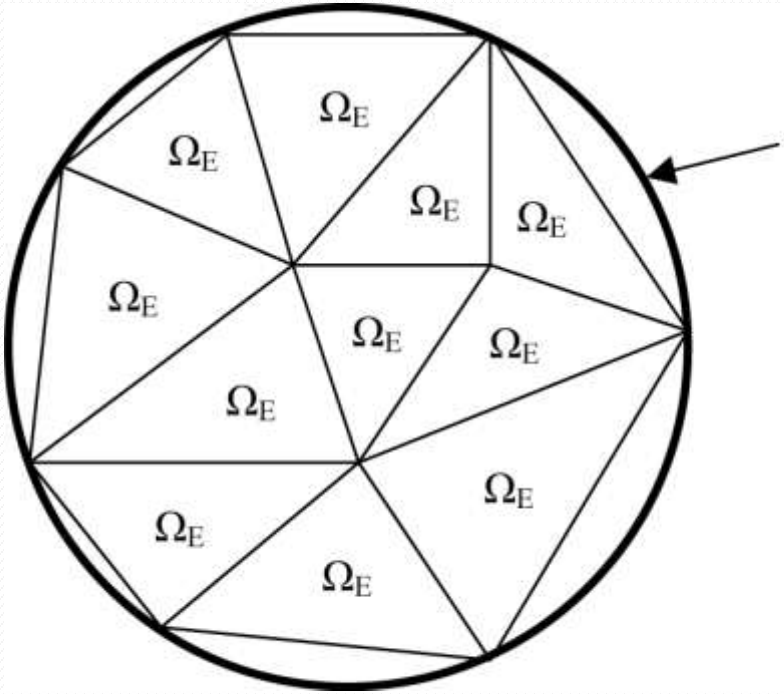
It works! ----- FEM

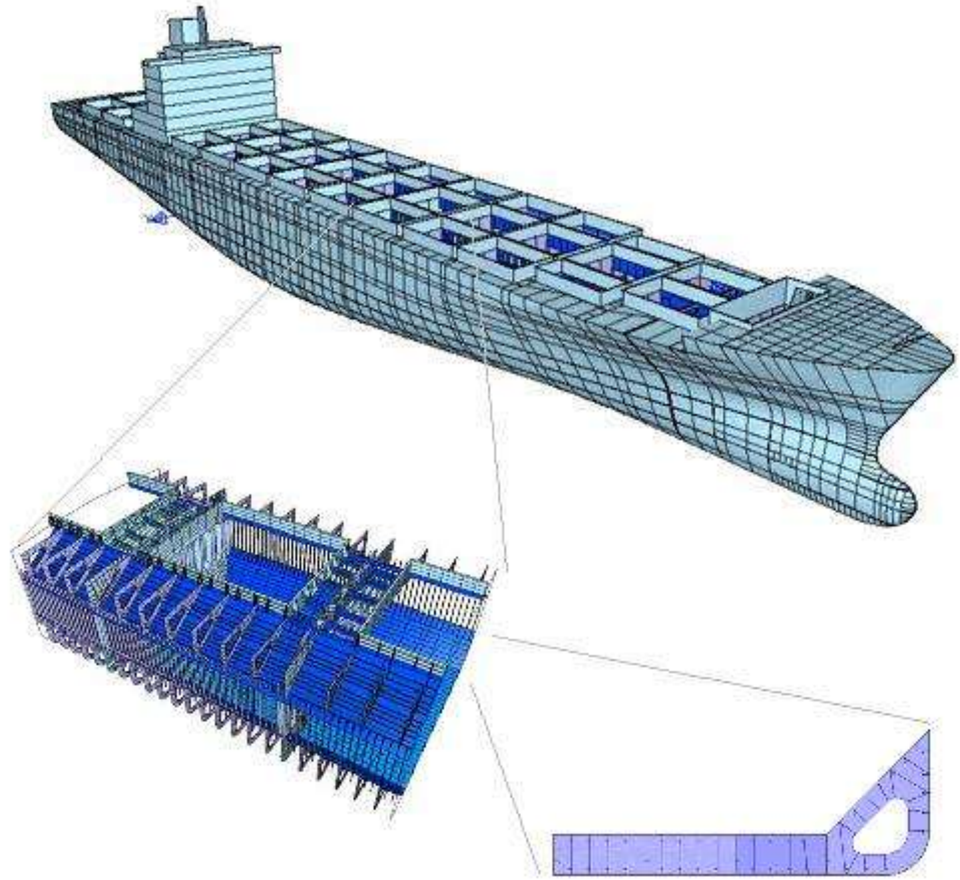


FOS = 1.103!

It does not work! ----- LEM

What is the real story?





Question:

What is an Element?

An Element and the Interface

- What is an “Element”?
  - A division of the domain. An atom. The smallest mechanical entity, where a constitutive relationship can be directly applied. An Element is a smallest mesh!
- Types of elements
  - 1D, 2D or 3D elements (A FE program has a library of elements)
  - Solid elements, shell elements, beam elements, etc
- **Connections between elements**
  - The “connection” between finite elements shall be clearly understood (particularly true when the connection is between different type of elements)
  - The interface behavior shall be clearly understood (sliding, friction, contact, separation etc)

Part 2: How Does It Work ?

The FEM Program

- Input (Pre-processing)
- Solution (Core [Black Box])
- Output (Post-processing)

User

More mutual understanding?

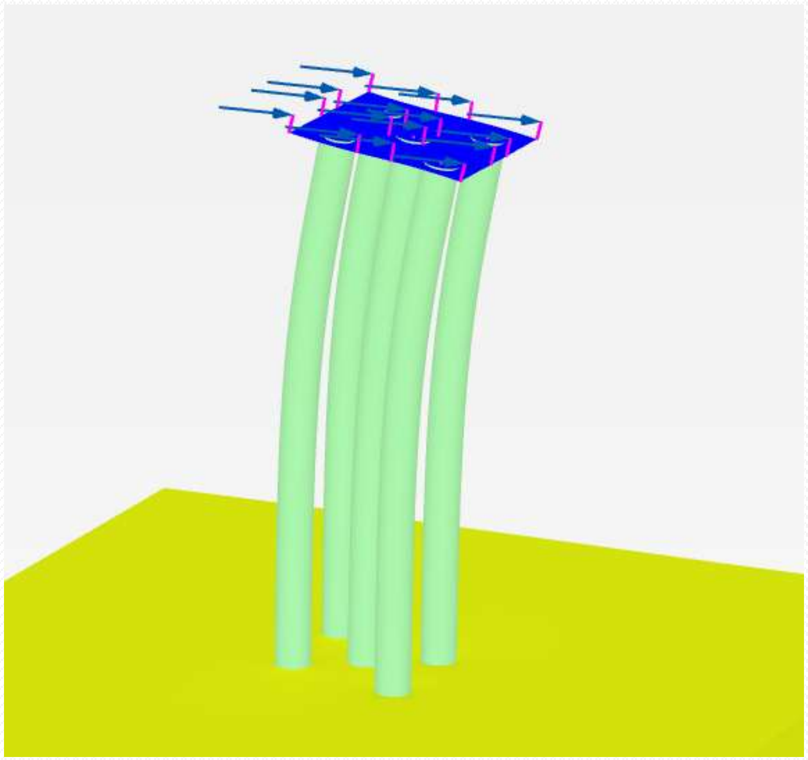
Developer

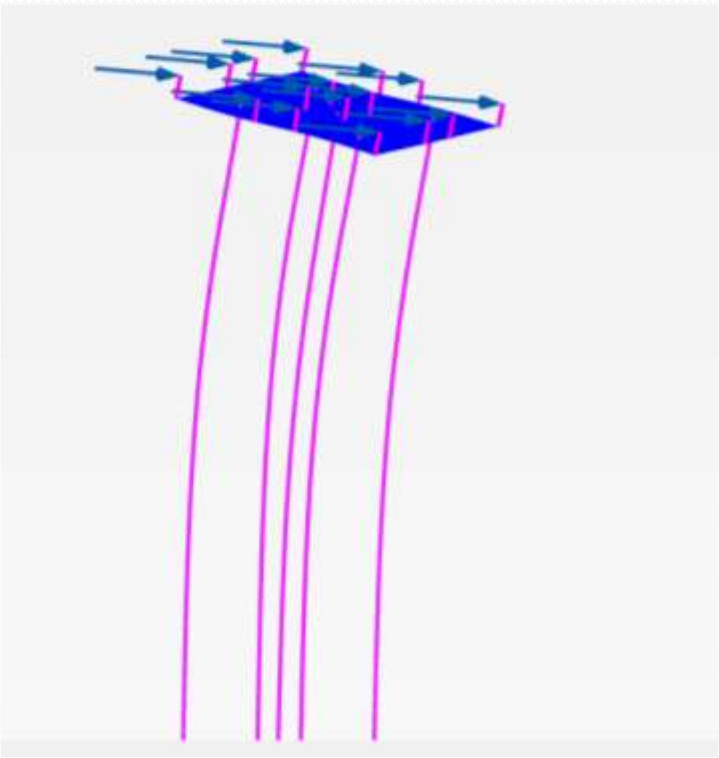
Input

- Geometry
- Loadings and boundary conditions
  - Loadings include body force and surface force
  - Surface force is one kind of boundary condition
- Material properties
- Solution control parameters

The problem shall be “well-defined” !

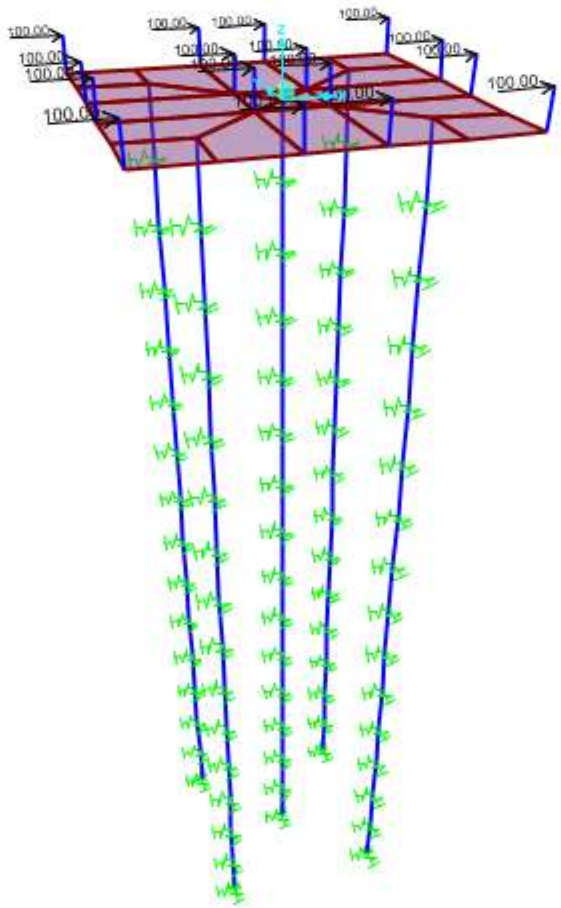
Are they equivalent??





Pile Modeling:
“embedded pile”, “beam” or solid element?

How about this?



Simplifications Can Lead to Troubles

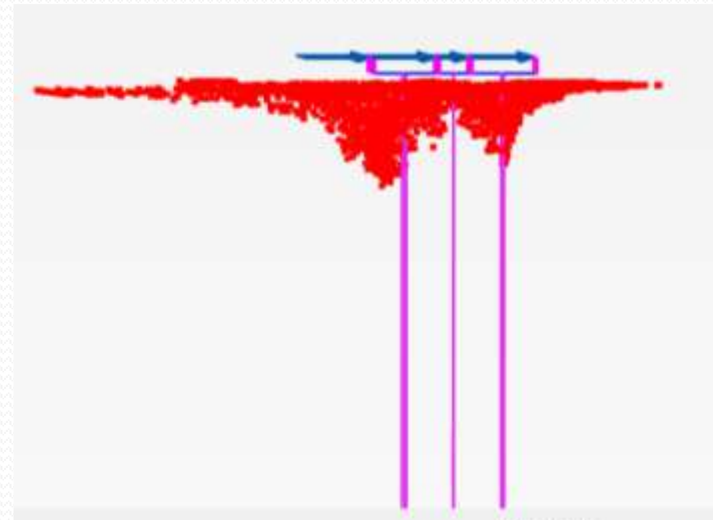
- The following are common in design but “new” or “difficult” to finite element program
  - Slip surface
  - Sub-grade modulus
  - Poisson’s ratio =0.5 for saturated clays
  - Factored strength and modulus parameters
  - Point loads
  - ……

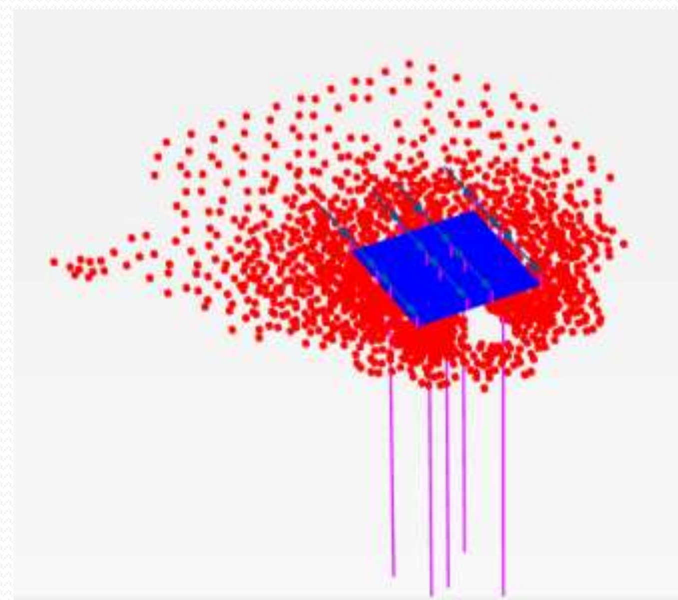
Don’t take your design-world experience for granted. Examine your assumptions.

Output

Output:

- Direct output: unknown degree of freedom, typically nodal displacements
  - Average at nodes or elements: continuous or discontinuous contour plots
- Derived quantities: typically stresses, strains, moments and forces
  - e.g. stress points vs. nodes





- Verify your results
  - Visual inspection of displacement contours
  - Checking on a few key indicators
    - Against hand calculation
    - Alternative methods
  - Peer review

Solution Process

- Solution algorithm
  - Implicit algorithm
    - ANSYS, ABAQUS-Implicit, SAP2000, PLAXIS
  - Explicit algorithm
    - LS-DYNA, ABAQUS-Explicit

Solution Process for Implicit Method

Time/Load Step

- Iteration loop
- Assemble force vector and stiffness matrix

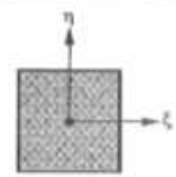
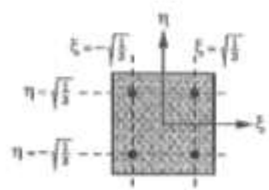
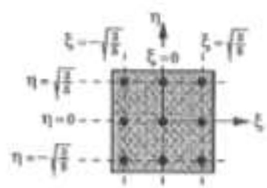
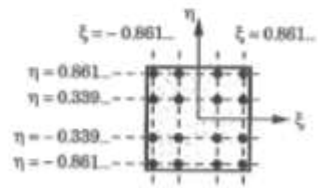
Loop over element

- Loop over Gauss integration point
  - Call material subroutine
- End loop

End loop

Next Time/Load Step

Table 9.3.1 Selection of the integration order and location of the Gauss points for linear, quadratic, and cubic quadrilateral elements (nodes not shown).

Element type	Maximum polynomial degree	Order of integration ($r \times r$)	Order of the residual	Location of integration points* in master element
Constant ($r = 1$)	0	1×1	$O(h^2)$	
Linear ($r = 2$)	2	2×2	$O(h^4)$	
Quadratic ($r = 3$)	4	(3×3)	$O(h^6)$	
Cubic ($r = 4$)	6	(4×4)	$O(h^8)$	

*See Table 6.1.2 for the integration points and weights for each coordinate direction.

For $l = 1, \ldots, n_{\text{int}}$

Set up $\widetilde{\boldsymbol{D}}$.

For $b = 1, \ldots, n_{en}$

Let $B_1 = N_{b,x}$ and $B_2 = N_{b,y}$.

Multiply $\widetilde{\boldsymbol{D}} * \boldsymbol{B}_b$, taking account of zeros in $\widetilde{\boldsymbol{D}}$ and $\boldsymbol{B}_b$:

$$\widetilde{\boldsymbol{D}}\boldsymbol{B}_b = \begin{bmatrix} \widetilde{D}B_{11} & \widetilde{D}B_{12} \\ \widetilde{D}B_{21} & \widetilde{D}B_{22} \\ \widetilde{D}B_{31} & \widetilde{D}B_{32} \end{bmatrix} = \begin{bmatrix} \widetilde{D}_{11}B_1 & \widetilde{D}_{12}B_2 \\ \widetilde{D}_{12}B_1 & \widetilde{D}_{22}B_2 \\ \widetilde{D}_{33}B_2 & \widetilde{D}_{33}B_1 \end{bmatrix}$$

For $a = 1, \ldots, b$

Let $B_1 = N_{a,x}$ and $B_2 = N_{a,y}$.

Multiply $\boldsymbol{B}_a^T * (\widetilde{\boldsymbol{D}}\boldsymbol{B}_b)$, taking account of zeros in $\boldsymbol{B}_a$, and accumulate in $\boldsymbol{k}^e$:

$$\begin{bmatrix} k^e_{2a-1,2b-1} & k^e_{2a-1,2b} \\ k^e_{2a,2b-1} & k^e_{2a,2b} \end{bmatrix} \leftarrow \begin{bmatrix} k^e_{2a-1,2b-1} & k^e_{2a-1,2b} \\ k^e_{2a,2b-1} & k^e_{2a,2b} \end{bmatrix} + \begin{bmatrix} (B_1\widetilde{D}B_{11} + B_2\widetilde{D}B_{31}) & (B_1\widetilde{D}B_{12} + B_2\widetilde{D}B_{32}) \\ (B_2\widetilde{D}B_{21} + B_1\widetilde{D}B_{31}) & (B_2\widetilde{D}B_{22} + B_1\widetilde{D}B_{32}) \end{bmatrix}$$

Gaussian quadrature of order N for the standard quadrilateral element

$$\int_{-1}^{1}\int_{-1}^{1} g(\xi,\eta)\, d\xi d\eta \approx \sum_{i=1}^{N}\sum_{j=1}^{N} w_i w_j g(\xi_i,\xi_j).$$

Example: *Gaussian quadrature of order 3 for the standard quadrilateral element $R_{st} = [-1,1]^2$:*

$$\begin{aligned}\int_{-1}^{1}\int_{-1}^{1} g(\xi,\eta)\, d\xi d\eta \approx & \frac{5}{9}\cdot\frac{5}{9}\cdot g\left(-\frac{\sqrt{3}}{\sqrt{5}}, -\frac{\sqrt{3}}{\sqrt{5}}\right) + \frac{8}{9}\cdot\frac{5}{9}\cdot g\left(0, -\frac{\sqrt{3}}{\sqrt{5}}\right) + \frac{5}{9}\cdot\frac{5}{9}\cdot g\left(\frac{\sqrt{3}}{\sqrt{5}}, -\frac{\sqrt{3}}{\sqrt{5}}\right) \\ & + \frac{5}{9}\cdot\frac{8}{9}\cdot g\left(-\frac{\sqrt{3}}{\sqrt{5}}, 0\right) + \frac{8}{9}\cdot\frac{8}{9}\cdot g(0,0) + \frac{5}{9}\cdot\frac{8}{9}\cdot g\left(\frac{\sqrt{3}}{\sqrt{5}}, 0\right) \\ & + \frac{5}{9}\cdot\frac{5}{9}\cdot g\left(-\frac{\sqrt{3}}{\sqrt{5}}, \frac{\sqrt{3}}{\sqrt{5}}\right) + \frac{8}{9}\cdot\frac{5}{9}\cdot g\left(0, \frac{\sqrt{3}}{\sqrt{5}}\right) + \frac{5}{9}\cdot\frac{5}{9}\cdot g\left(\frac{\sqrt{3}}{\sqrt{5}}, \frac{\sqrt{3}}{\sqrt{5}}\right).\end{aligned}$$

Part 3: Why Does It Work?

Basis of Theoretical Background

- Background of FEM
  - The link between FEM, Mechanics (continuum mechanics) and material science (constitutive laws)
- Governing equations
  - Balance laws
    - Balance of mass
    - Balance of linear momentum
  - Stress-strain relationship
    - Linear
    - Nonlinear

Governing Equations

- Strong Form $$\boxed{\sigma_{ij,i} + f_i = 0}$$

What engineers normally know

- Weak Form $$\boxed{\int w_{i,j}\sigma_{ij} d\Omega = \int w_i f_i d\Omega + \int w_i h_i d\Gamma}$$

- Matrix Form $$\boxed{\begin{aligned} &Kd = F, \\ &where\ K = \int BDBd\Omega \end{aligned}}$$

What the software know

$$\boxed{\{Strong\ Form\} \Leftrightarrow \{Weak\ Form\} \approx \{Matrix\ Form\}}$$

Hidden Assumption on Problem Continuity

Discretion Error

Part 4: What Else Can Go Wrong?

- Nonlinearity
- Convergence
- Stability
- Accuracy

Nonlinearity

- Nonlinearity
  - Geometric nonlinearity
  - Material nonlinearity
  - Contact nonlinearity

Received less attention from practitioners in Hong Kong

What engineers are normally concerned with

Including interface issues

All linear problems are trivial; and all nonlinear problems are impossible." R. Sachs Berkeley 1970
Author of: General Relativity for Mathematicians

Convergence

- Convergence
  - Mesh independent solution (coarse mesh vs. fine mesh)
  - Increase of the number of loading step or time step
- General criterion
  - The smaller the mesh, the accurate the solution (!not always true!)
  - The smaller the time step, the accurate the solution (!not always true!)

Stability

- An "insensitivity" to small perturbations
- Perturbations are modeling errors of systems, environment, noise
- For dynamic analysis: "stability requirement" implies a limitation on time step and element size

Developers

Users

- Stability of solutions to discrete equations
- Stability of time integration procedures
- Material stability (softening)

Developers

Users

Accuracy

- First order accurate
- Second order accurate

Mathematical form

$$e^{(i+1)} = A \times e^{(i)},\ \ A < 1 \Leftrightarrow \text{unconditionally stable}$$

$$e^{(i+1)} \leq c\,|e^{(i)}|^{k},\ \ \ k = \text{rate of convergence/accuracy}$$

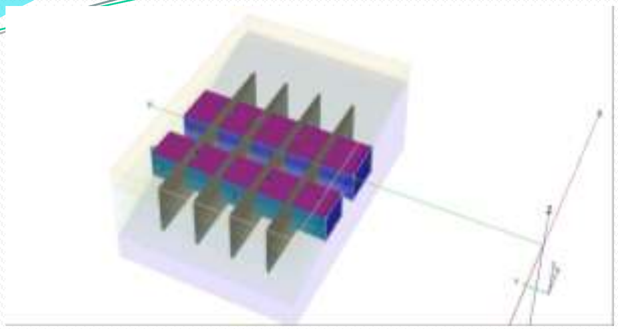
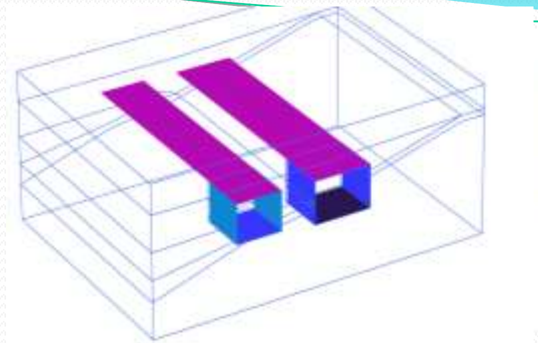
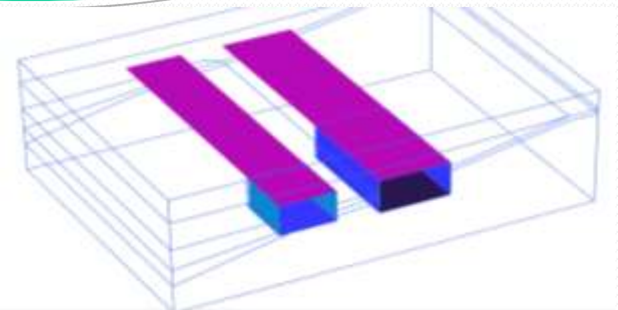
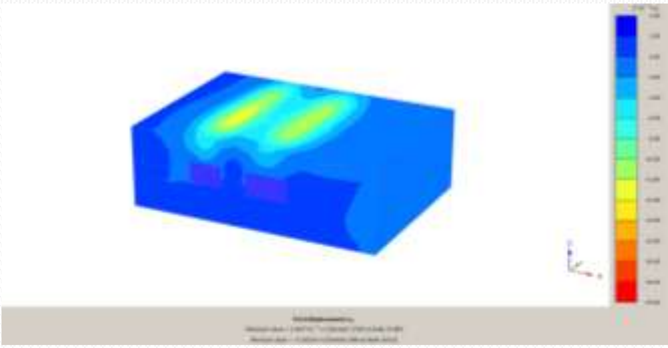
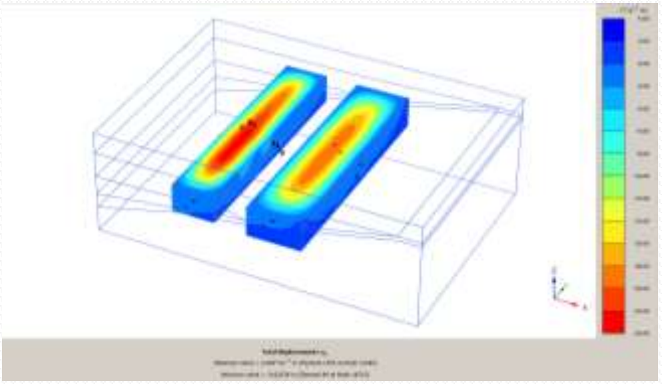
Why FEM is NOT accurate for geotechnical problems

- Uncertainty in material behavior
  - One-phase material vs. two-phase material
  - Limited geological information
  - Derivation of parameters based on empirical rules
  - Highly nonlinearity (3 types!)
- Uncertainties/simplifications in Loading, Boundary Conditions and Geometry

Part 5: Back to the Analyst's World

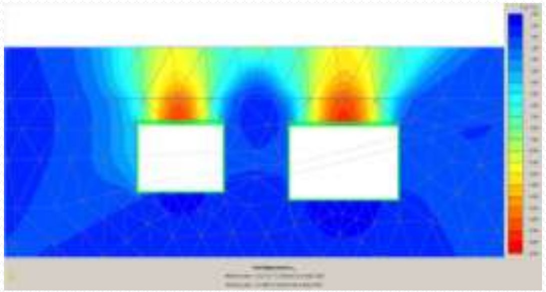
- Geometry/Mesh
- Material
- Loadings
- (Initial) boundary conditions
- Convergence, stability
- Interpretation of results

Only an assembly of all of the above generates a valid solution!

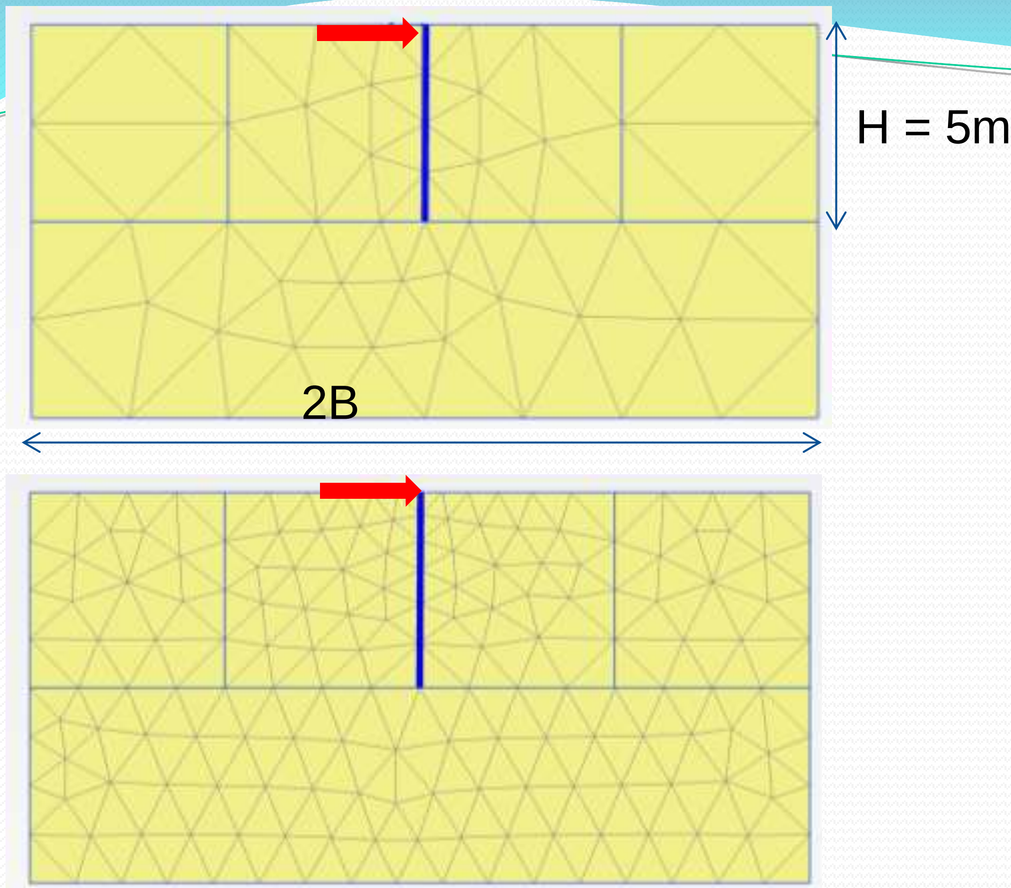
Analysis: 3D

How to check whether the modeling is sufficient?



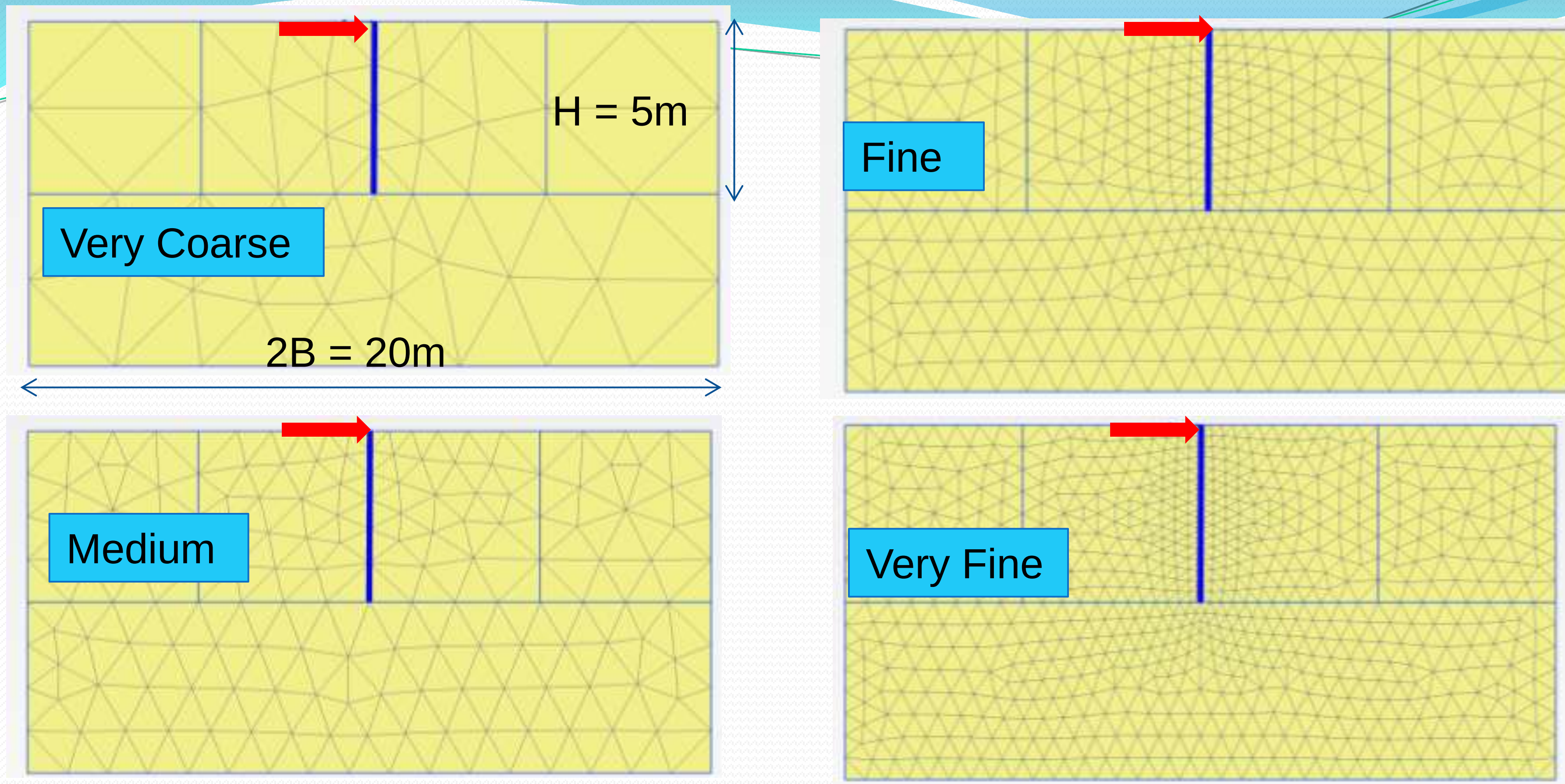
Design: 2D

How to ensure design is robust?



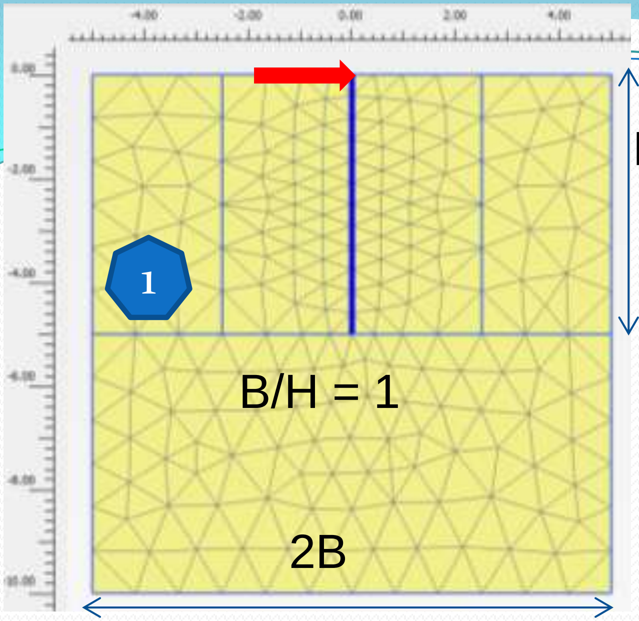
Mesh Coarseness	Delta	Difference
Very Coarse	22.72	0.6%
Medium	22.85	0.0%

Good Agreement! Is it enough?



Mesh Coarseness	Delta	Difference
Very Coarse	22.72	4.3%
Medium	22.85	3.7%
Fine	23.34	1.7%
Very Fine	23.74	0.0%

1 mm Improvement!
Is it enough?



B/H	Delta	Difference
1	19.03	27.0%
2	22.85	12.4%
3	24.74	5.1%
5	25.63	1.7%
10	26.07	0.0%

To the FE Program

Problems 1,5, and 10 are different BVP (boundary value problems)

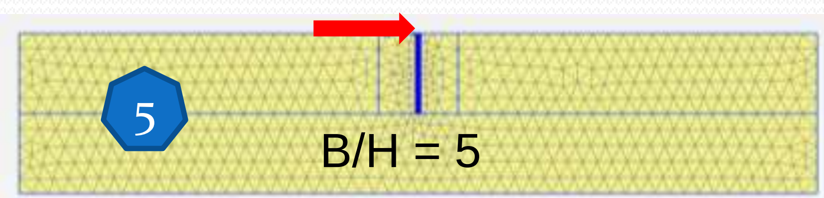
To the analysts/users, they may be used to simulate the same physical problems.

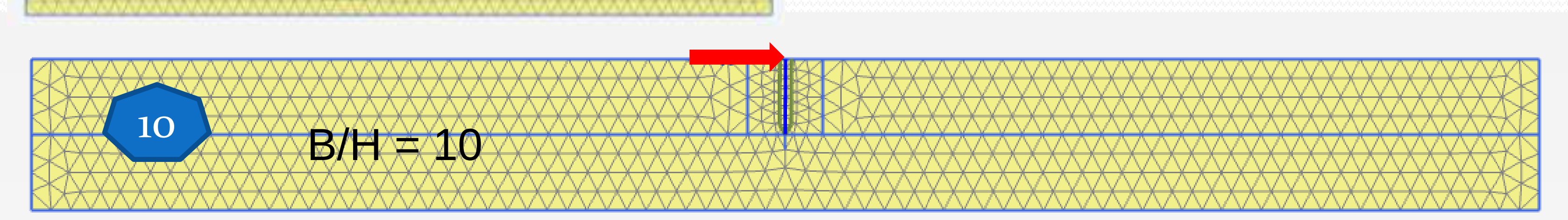
The physical reality of the physical problems may NOT be Case 10.

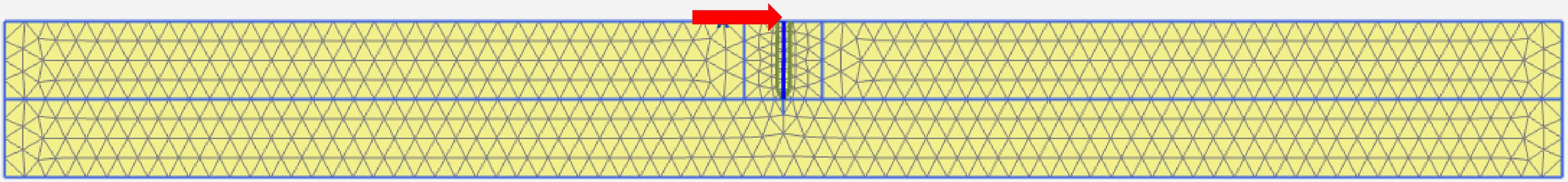
B/H = 2

B/H = 3

Convergence Doesn't Mean Accurate!



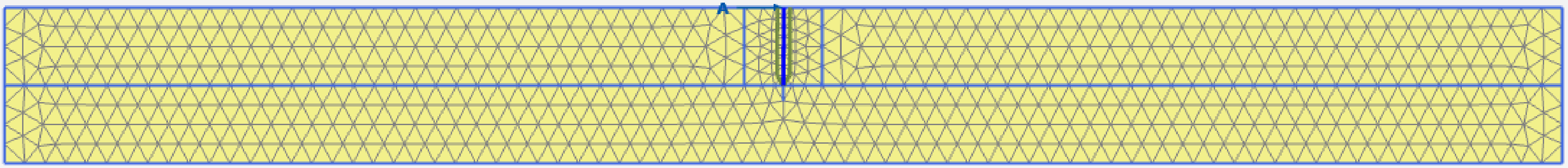




Without interface: max soil disp = 26.07mm

Interface factor	Delta	Difference
1	29.78	-14.2%
0.9	30.17	-15.7%
0.7	30.55	-17.2%
0.5	32.21	-23.6%
0.2	36.35	-39.4%
0.1	53.04	-103.5%

Something WRONG with the Interface?

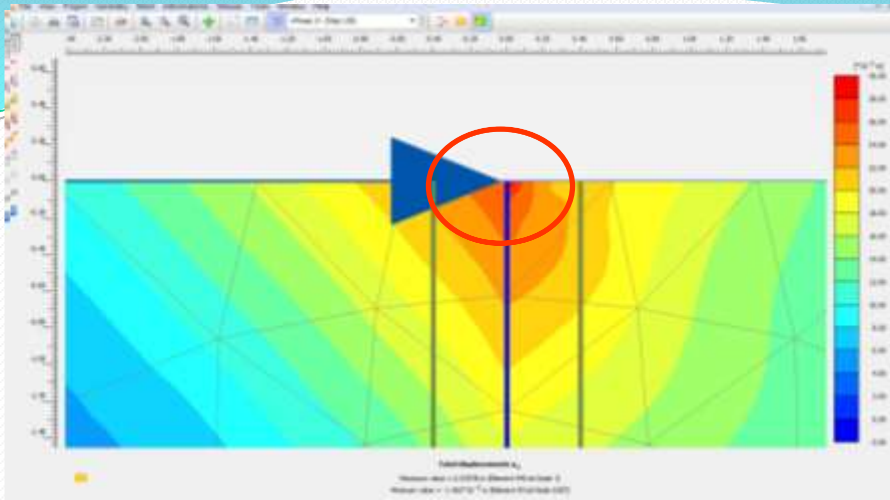


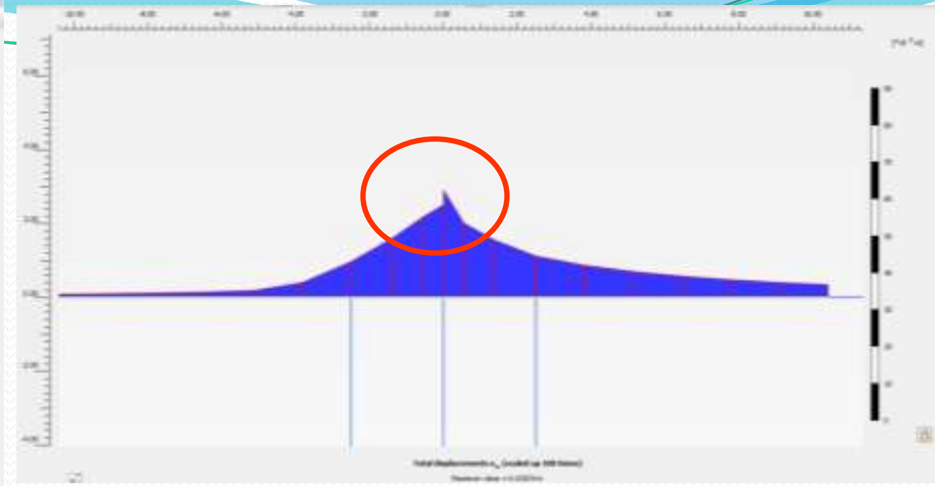
Without interface: max pile disp = 26.07mm

Interface factor	**Delta (Pile)**	Difference
1	26.03	0.2%
0.9	26.19	-0.5%
0.7	26.70	-2.4%
0.5	27.92	-7.1%
0.2	36.00	-38.1%
0.1	51.51	-97.6%

Interface factor	**Delta (Soil)**	Difference
1	29.78	-14.2%
0.9	30.17	-15.7%
0.7	30.55	-17.2%
0.5	32.21	-23.6%
0.2	36.35	-39.4%
0.1	53.04	-103.5%

Why is there a difference between maximum Soil and Pile response?





Interface displacement plot

Wall displacement plot

The difference is related to the element itself !

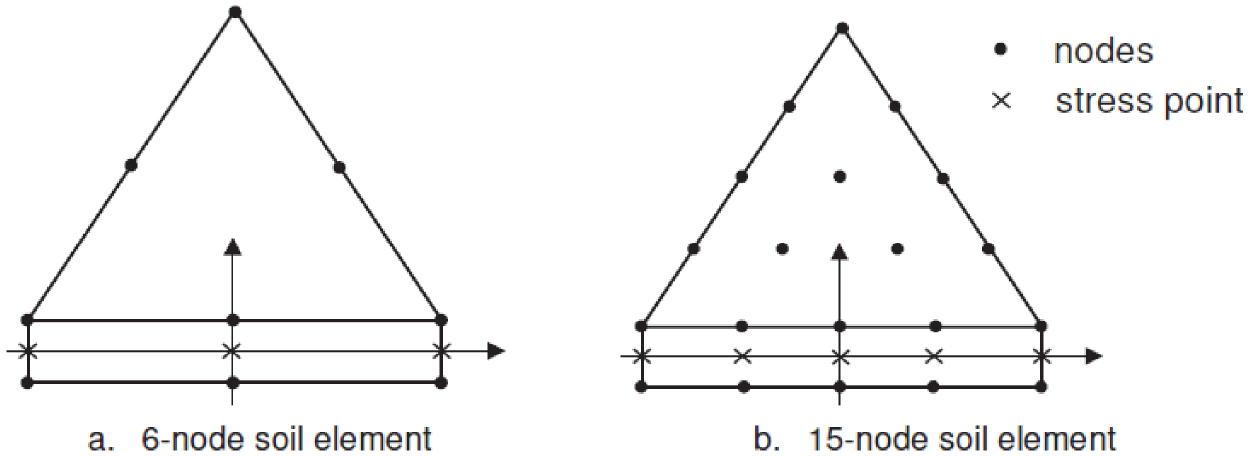


Figure 3.19 Distribution of nodes and stress points in interface elements and their connection to soil elements

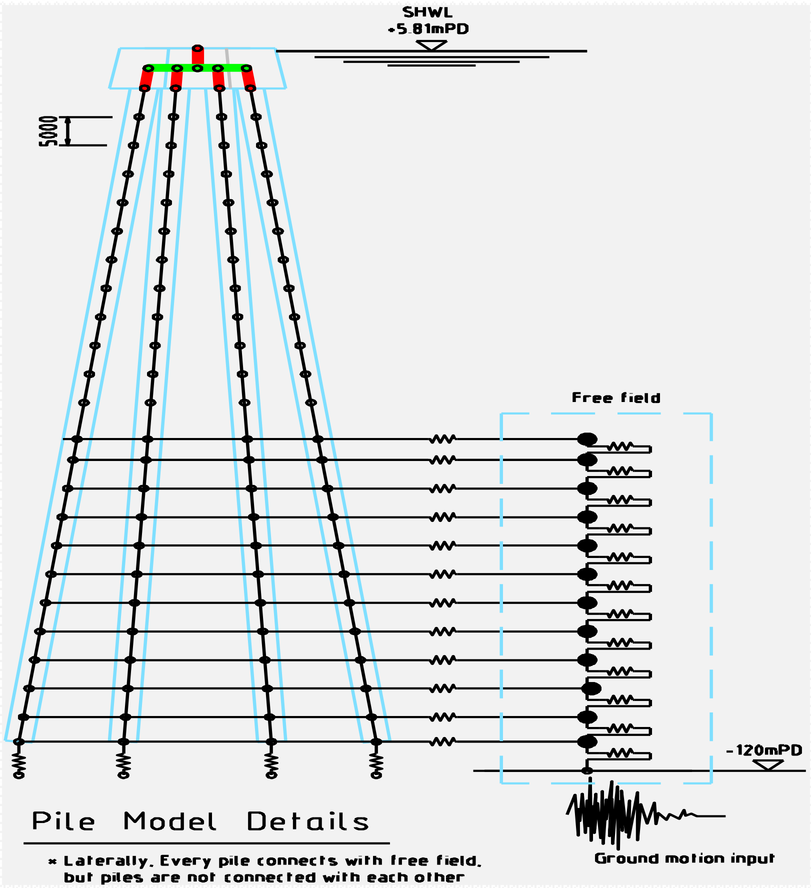
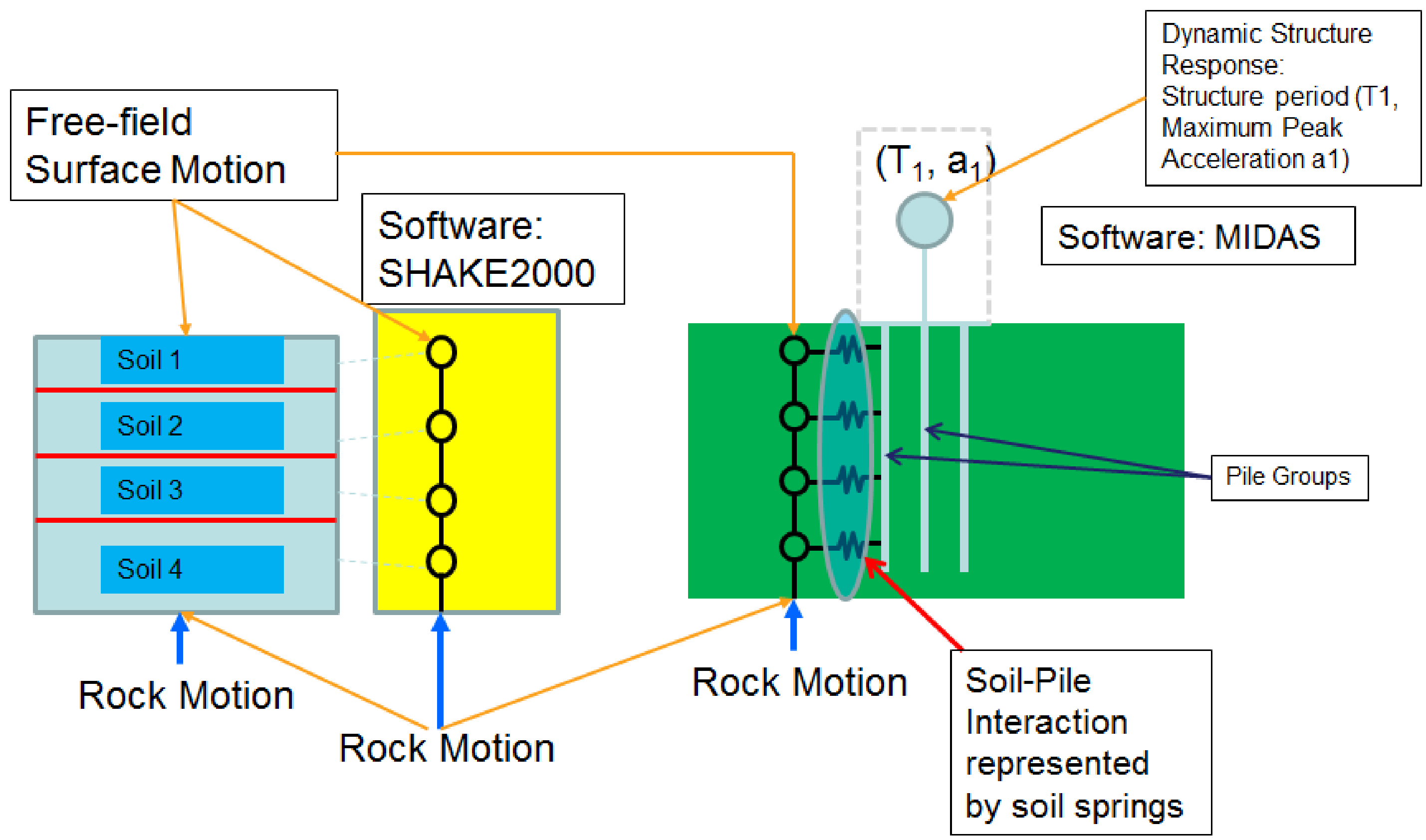
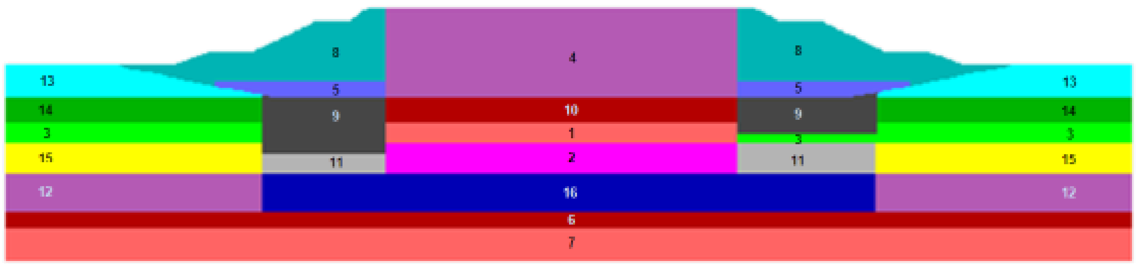

SHWL
+5.81mPD
5000
Free field
-120mPD
Pile Model Details
* Laterally. Every pile connects with free field.
but piles are not connected with each other
Ground motion input

Illustration of the Soil-Pile-Structure System





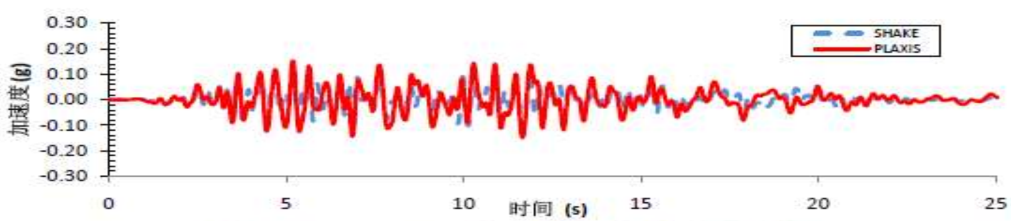
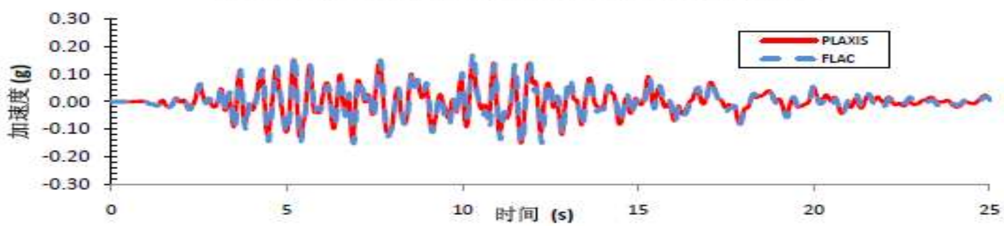
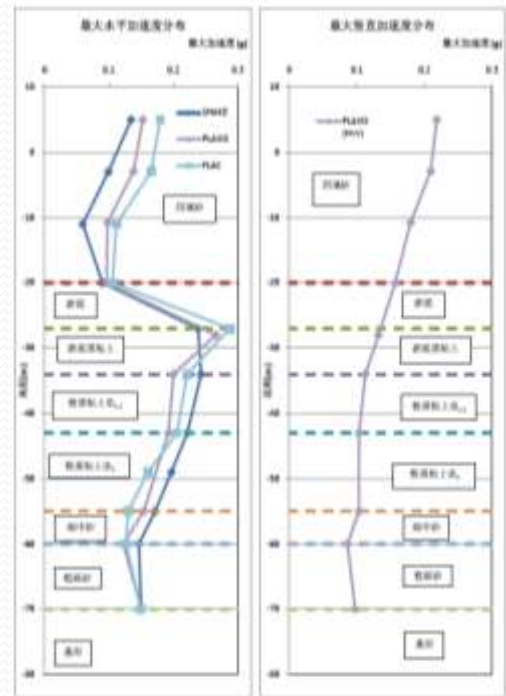
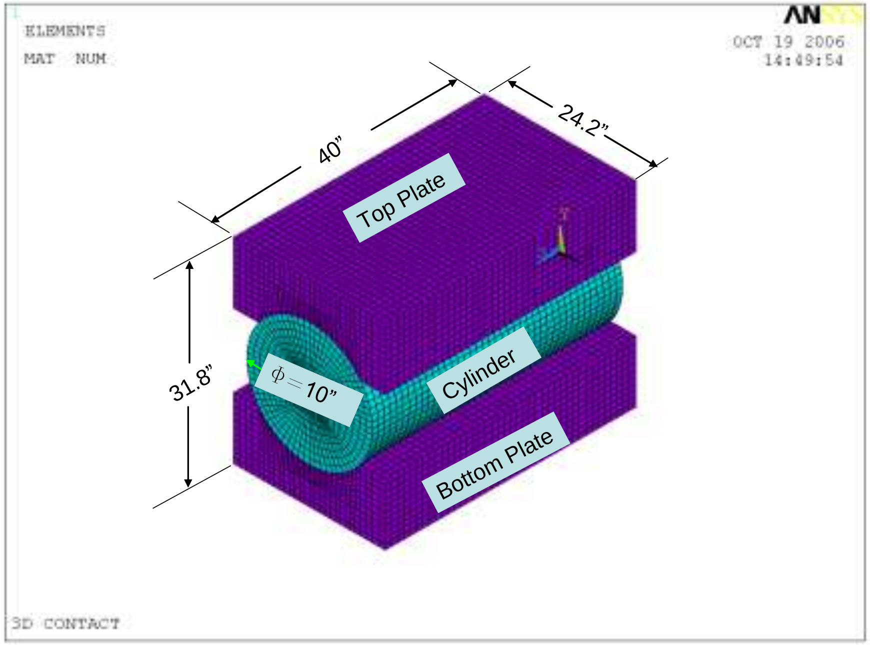
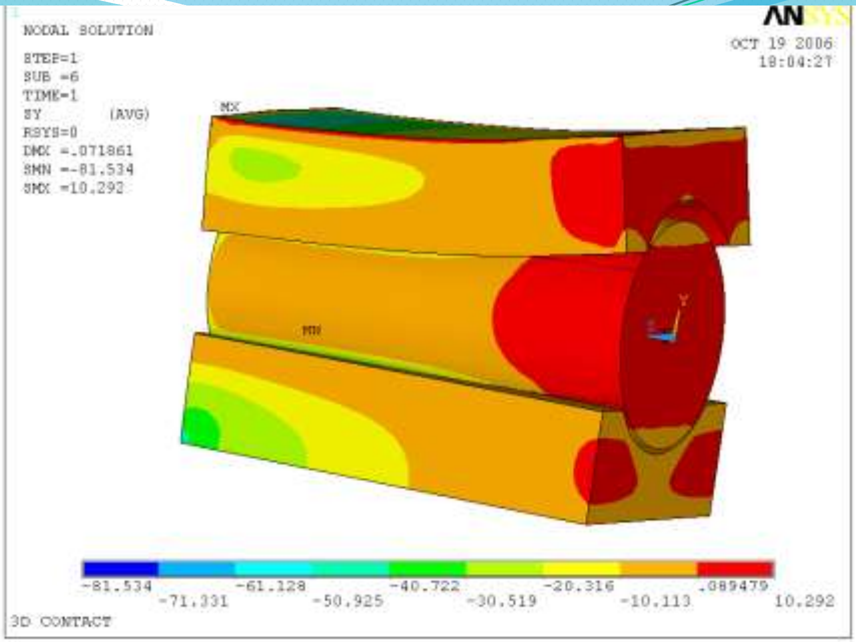


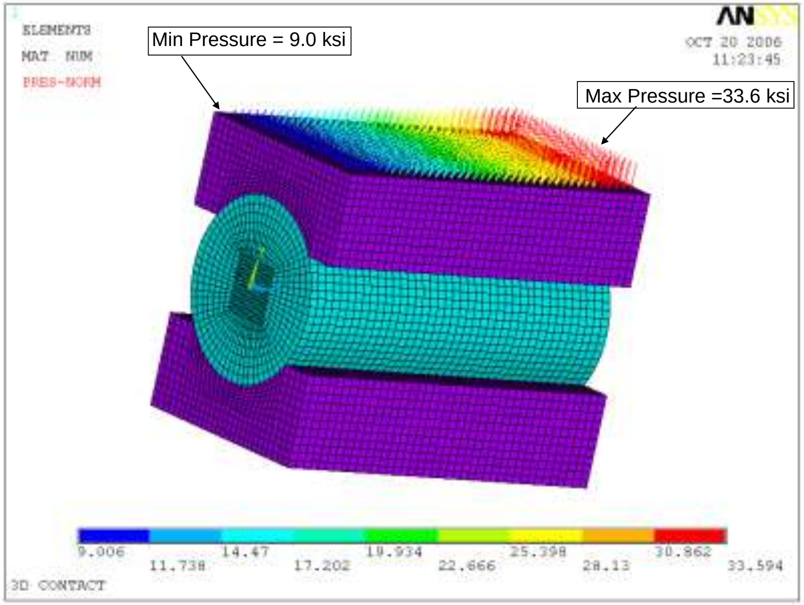
图 5-78 PLAXIS、SHAKE 两种软件地表水平加速度时程曲线对比

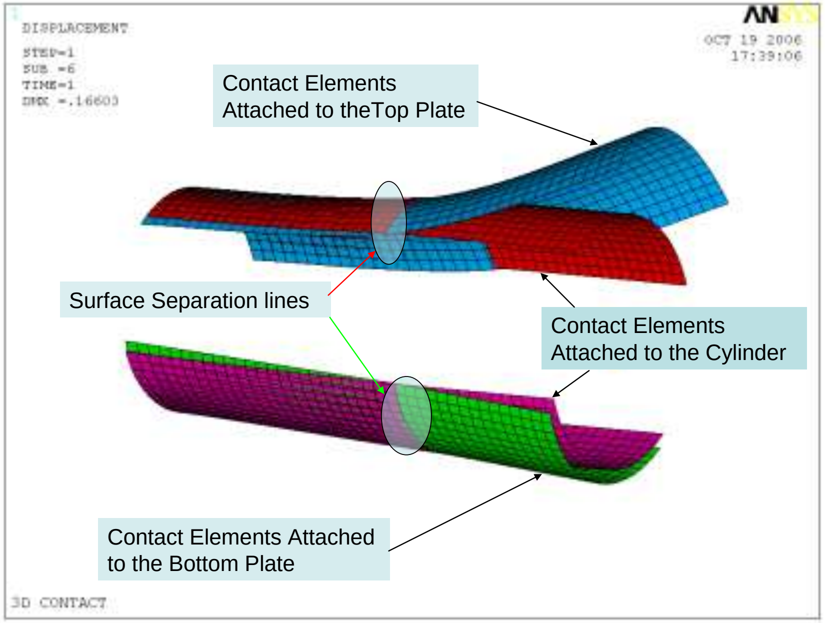




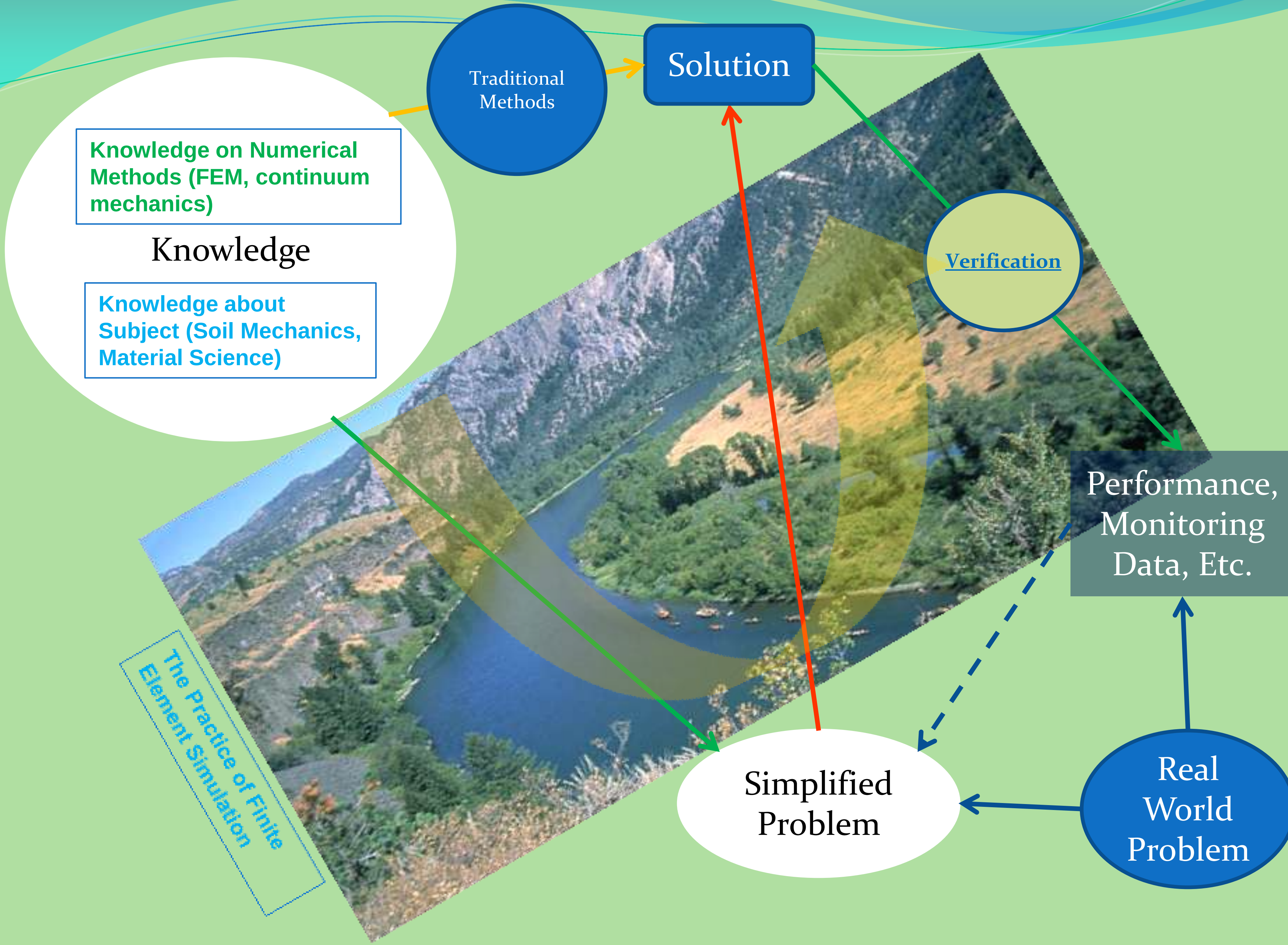

ELEMENTS
MAT NUM
OCT 19 2006
14:49:54
40”
24.2”
Top Plate
31.8”
ϕ = 10”
Cylinder
Bottom Plate
3D CONTACT


NODAL SOLUTION
STEP=1
SUB =6
TIME=1
SY (AVG)
RSYS=0
DMX =.071861
SMN =-81.534
SMX =10.292
OCT 19 2006
18:04:27
MX
MN
-81.534
-71.331
-61.128
-50.925
-40.722
-30.519
-20.316
-10.113
.089479
10.292
3D CONTACT


ELEMENTS
MAT NUM
PRES-NORM
Min Pressure = 9.0 ksi
Max Pressure =33.6 ksi
OCT 20 2006
11:23:45
9.006
11.738
14.47
17.202
19.934
22.666
25.398
28.13
30.862
33.594
3D CONTACT


DISPLACEMENT
STEP=1
SUB =6
TIME=1
DMX =.16603
OCT 19 2006
17:39:06
Contact Elements
Attached to theTop Plate
Surface Separation lines
Contact Elements
Attached to the Cylinder
Contact Elements Attached
to the Bottom Plate
3D CONTACT

Numerical Modeling and It's State of Practice



Study the Software!

- Study the tutorial and examples provided in the manual
- Run benchmark problems
- Sanity check
- Hand calculation to know the cause-effect relationship
- Use common sense and engineering sense
  - Don't believe in the accuracy of the displacement contour if your material model is not right or the parameters are off-limit

For Engineers in Charge

- Form a team with both analytical skills and engineering experience
- Give enough time to the analyst, start as early as possible
- Try to involve experienced engineers early in the modeling phase, especially when deciding loading, boundary conditions, material models
- Make sure the analyst has gone through the benchmark problems before tackling a real-world problem
- Understand the limitation of the simulation results and its usage

Concluding Remarks:

- Be aware, be alert and be cautious

When to ask for help?

- Not sure what loading to apply
  - Discuss with experienced engineer to find out the correct magnitude
  - Make sure the way you apply loading doesn't affect the results adversely (line loading, point loading, pressure loadings etc)
- Not sure of how many stages are needed
- Be aware of what parameters are used
  - Perform some kind of sensitivity study if time permitted

Acknowledgement

- Thomas J.R. Hughes: The finite element method
- J Bonet and R Wood: Nonlinear continuum mechanics for finite element analysis
- J.C. Simo and T.J.R Hughes: Computational inelasticity
- PLAXIS, ANSYS, SHAKE2000, FLAC, SAP2000, and MIDAS
- Internet Photos obtained on Ship and Airplane examples
- Current and previous employers
  - Lambeth Associates (A Member of Gammon Group)
  - URS Benaim
  - AECOM Asia
  - Weidlinger Associates

Thank You!

Questions? Comments?

chao.li@lambeth.com.hk
chaoli@gmail.com

Lambeth Associates Limited
(A member of Gammon Group)
28/F Devon House, Taikoo Place, 979 King's Rd, Hong Kong
Tel: (852) 2516 8688   Fax: (852) 2516 6352