ABAQUS

→ Abaqus FEA is a software suit for FEA originally released in 1978. This suit has 5 core software products.

- Abaqus/CAE (complete abaqus environment computer aided engineering)
- Abaqus/Standard (An FEA analysis with implicit integration scheme)
- Abaqus/Explicit (for highly NL systems with complex contacts under transient loads)
- Abaqus/CFD (computational fluid dynamics)
- Abaqus/Electromagnetic

→ Abaqus use "Python" scripting language.

→ Modeling levels (Nonlinear)

Lumped plasticity → Fiber modeling → FEA

(interested in local responses of small FEs)

→ ABAQUS
- → Material level modeling
- → Contact problems
- → Modeling temperature changes
- → " in " Electrical conductivity
- → Fluid modeling (liquid sloshing etc.)

→ FE Integration Scheme

Explicit

Good for dynamic Analysis, Small time step

Implicit

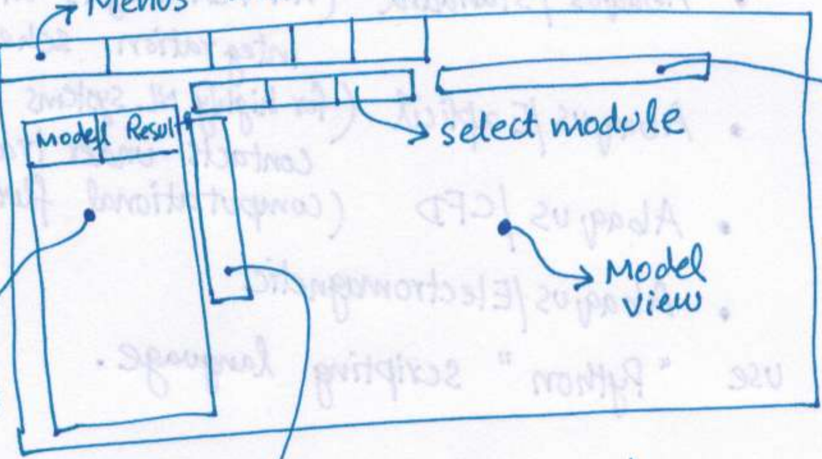
For "quasi-static" behaviors (You define allowable error), not efficient.

→ Abaqus have "Modules" → like a workshop.

→ Open Abqus/CAE

Create Model database
- Standard/Explicit
- CFD
- EM

Menus will change with module



Toolbars/Viewing options

select module

Model view

To navigate b/w modules and their options (whole detail of model shown)

Commonly used options/change with Module, can also be found in Menus above.

→ Modules
- → Part
- → Property
- → Assembly
- → Step
- → Interaction
- → Load
- → Mesh
- → Optimization
- → Job
- → Visualization
- → Sketch

Abaqus is unitless → You have to make sure that units are consistent. Can google to know consistent units. e.g. If you are in N, mm, sec units, the mass should be in Tons and stress in MPa.

Example

→ Lets take an example, An RC (PC) column resting on a concrete isolated footing.

For this example, we need 2 "parts" — Column, Footing

and we will Join them in "Assembly" and will define their behavior of connection in "Interaction". We will define its loads in "Loads" module, its meshing in "Meshing module" and will define its analysis methods etc in "Step" module.

The logical sequence of all these tasks is already listed in left "Model database" → All important options regardless of what module they belong. Materials etc are in "Property" module.

→ To create a new part → Double click on "Parts" in left side "Model Database".

- Enter part name " e.g "Column"
- 3D
- Deformable
- Solid (Shell for thin plates, wire for axial 1D behavior) + Extrusion with Solid
- Any approximate size
- Continue

A drawing interface will open with scripting language commands at bottom. Draw rectangle ↓ Click ⌜⌟ and give 2 points (0,0) enter (300,300) enter then $\boxed{X}$ → Done

- Sketch → done

With "Done", a new window will open, asking for depth of 3D part.
give "Depth = 3000". This is extrusion. → OK

→ A column with size 300 x 300 x 3000 will be formed.
↳ part

→ Repeat this process to again create a "Part" named foundation (1000 x 1000 x 300) (we are following mm length units).

→ Under each new Part, Following options appear in "Model database"

- Features
- Sets
- Surfaces
- Skins
- Stringers
- Section assignments
- Orientations
- Composite layups
- Engineering features
- Mesh (empty)

→ Double Click "Materials" in Model Database.
Or Go to "Property" Module and click

in right toolbar, to create a new material.

Name = Concrete

Properties
- General
- Mechanical
- Thermal
- EM
- Others

- General → Density
- Mechanical $\xrightarrow{\text{if}}$ Elasticity → Elastic → give E, ν
  - → Plasticity → Concrete Damaged Plasticity (For NL Behavior)
    - Plasticity
    - Comp behavior
    - Tensile Behavior
      - fy
      - Cracking strain
- Add all required "Material behaviors"
- OK

→ Double click on "sections" or Select "Property" module and click in Toolbar.

- Name = Column
- Solid → Homogeneous
- Continue → give material
- OK

→ Go again to "Column" part → "Section Assignments" Select the section with click → OK → Select what Section to assign this part. → OK. Part color will change to green.

→ "Sets" and "surfaces" are used, if you want to get results for a group of elements or nodes, assign boundary Conditions or constraints etc. etc. It is like "Add Frames" option in PERFORM 3D. (e.g if you want shear at a section passing through elements at column base)

→ In order to do Meshing, first we have to do "Seeding". Its like marking to cut later.

Seeding is to define shape of mesh and size.

→ Go to "Mesh" module. Click ~~Mesh~~ Seed menu on top or check the mesh side toolbar.

Mesh
- Part (If you want to give same size of each element)
- Edges (If you want cut along a particular edge)
- Delete part seeds
- Delete edge seeds

• Part → Just give global size of element

• Edge → Select any edge of column and done.
- By Size — give app size
- By Number — give no of elements

Bias
- None → If same size of elements along edge.
- Single, Double → If variable sizes along edge (single means  and double means 
  → For "By Size", give max and min sizes. For "By Number", give No. of elements and Bias ratio.

Flip bias = To change the direction of size change, along the edge.

"Mesh" Menu →
- Controls
  - Hex
  - Structured
- Element Type
  - Element library → Family
  - Geometric Order
  - Hex, Wedge, Tet

Family = 3D stress , Standard (Implicit , Explicit

Order = Linear

Implicit → Uses the output of previous Step. High margin of error. Have to specify Tolerance.

Explicit → Direct, each step is complete in itself. good for dynamic analysis, fast.

(For each part type, different element options will be shown)

Hex → ☐ Reduced Ingeratiteon → Hour-glass modes.

At bottom, the element description will be Shown, e.g. $\underbrace{\text{C3D8}}\underbrace{\text{R}}$

R → Reduced Integration

C3D8 → 8-noded linear brick

→ Menu Mesh → Part → OK (To actually divide the column in to C3D8R elements)

↓

Yes

→ Double click "Assembly. OR go to "Assembly" module → Click "Create Instance".

Instance is the copy of part.

- Parts
- select actual part (can select more than once)
- Independent
- ☑ Auto-offset
- OK

Use "Rotate Instance" and "Translate Instance" to put column on footing or to orient parts in to exact required assembly. No "interaction" have yet be defined, between two instances.

→ Double Click on "Interaction Properties"

Name:

Type : Contact → Continue

Contact property
- Mechanical
  - Tangential Behavior
  - Normal Behavior
- Thermal
- Electrical

Tangential Behavior (Sliding behavior)
- Frictionless
- Penalty (good option for solution convergence)
- Static-kinetic exponential decay
- Rough (No slip after contact)
- Langrange Multiplier.
- User-defined

In penalty, Friction coefficient = 0.5 for concrete-concrete.

Normal Behavior
- "Hard" Contact
- Exponential
- Linear
- Tabular
- Scale factor (General contact, Explicit)

"Hard" contact and penalty (Standard) are generally used.

→ Now we will create 2 "surfaces" to which we will assign this interaction created in previous step.

Go to Part (e.g column) → Double click "Surfaces" → Geometry → Select bottom face of column (which have to be in contact with footing) → done.

Make another surface of footing (its upper face) but use the option "By Mesh" → select only middle elements which will be in contact with column.

Create another surface of full bottom face of footing → To restraint it later.

→ Again go to "Interactions" → Double click

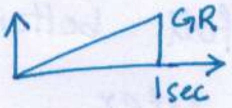
- Name = col-to-Footing
- Surface - to - surface contact (Explicit)
- Continue
- Select first surface
- $\boxed{\text{surface}}$ → Select 2nd surface
- Done
- First > which will tend to move into
  2nd / which
- Mechanical constraint — Kinematic contact
  └ Penalty
- Contact Interaction property → Select Your property.
- OK.

→ Double click on "BCs"

- Mechanical → Symmetry/Antisymmetry/Encastre
- Continue
- Select Surface → done
  ↳ bottom face of footing
- Encastre → All fixed
- OK

→ "Steps" are like load cases or analysis series. 1 step called "Initial" is already there by default.

That default step contains — Our defined Interactions and BCs.

e.g we want gravity load analysis, so we can add 1 more step giving it some time eg. 1 sec. 

. Then e.g we want quasi-static or dynamic analysis, we give 

.

→ Double click on "~~Sets~~ steps"

- Dynamic Implicit (e.g flexural analysis of column)
- Dynamic Explicit (contact, high-freq problems)
- static, General
- ⋮

e.g Dynamic, Explicit → Basic (Time period)
→ Incrementation (may be automatic)
→ Mass Scaling
→ Other

→ If you double click "Section sketch" in "Solid Extrude" of any "Features" of any "Part" → you will again have all drawing options. (e.g.  Add dimension)

In first session, we modeled a column and a footing. We will now continue with the same example.

→ In "Interactions" Module → we define the type of interaction between two surfaces (of instances).

"General Contact" is the most simplest type. It simply means that if two surfaces or objects come closer to eachother (closer by a certain small value), consider it as "contact".

Usually we use "Surface-to-surface Contact (Explicit)"

Select it and "continue" → Select the master surface (you have to create before, or can create now) + chose the slave surface. $\boxed{\text{Surface}}$ or $\boxed{\text{node region}}$

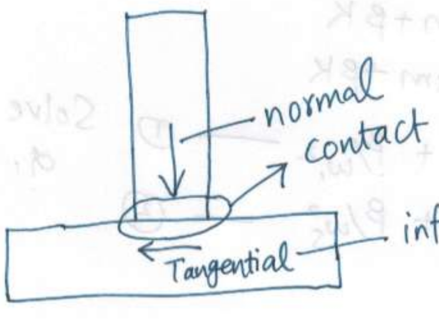
Done

Have to give "Interaction Property" here.

Go to "Interaction Properties" from here or from its module to define the "type" of contact.

→ Using "Contact" type, make a new interaction property. → Continue

Mechanical
- Tangential
- Normal
- Damping
- Damage
- ⋮

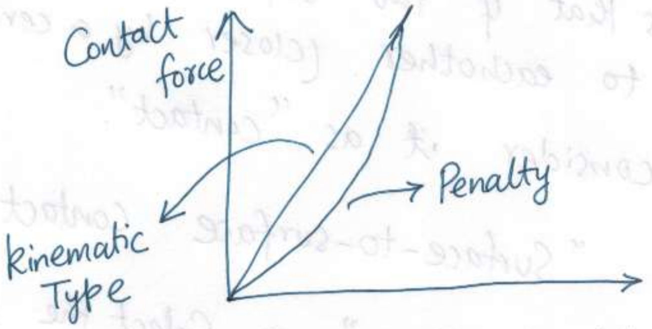


infact frictional behavior in our case.

→ Tangential Behavior ~~is~~ — Frictionless
- Penalty
- ...
- ...
- Rough

- frictionless is opposite of rough.
  ↓
  (Infinite friction)

Usually we use "Penalty". — You have to give 2 things:
- Coefficient of friction
- Shear stress limit (on which sliding will start)



→ Normal behavior → We will use "Hard Contact".

It means K of component A, " " B (automatically Calculated contact K) using individual Ks of both A and B.

If you select "Linear" → you have to give a direct contact stiffness.

In "hard Contact" → you can check "Allow separation after contact" → To model rocking behavior.

→ Modal Damping cannot be given in GUI, it can be given in input file.

→ In "~~Raleigh~~ Rayleigh Damping", you provide damping for 2 modes. eg M1 = 2.5% 10 HZ
M5 = 5% 50 HZ

C = αM + βK

$$2m\omega_1 = \alpha m + \beta K$$

$$2m\omega_5 = \alpha m + \beta K$$

$$2\omega_1 = \alpha + \beta/\omega_1^2 \quad \text{— (1)}$$

$$2\omega_5 = \alpha + \beta/\omega_5^2 \quad \text{— (2)}$$

Solve for α, β

→ Damping is provided in "materials" module

→ β is calculated from K (meshing etc) → Requires very high computational power. Some people just use $\beta = 0$ to save time.

In Raumoko, if you specify $\beta = 0$ → it considers αM as modal damping (option 4).

→ In "Materials" → Mechanical → Damping → α, β

→ In "Assembly" → Engineering features → Springs/Dashpots for external springs, dampers etc.

→ "STEPS"

Whatever we were doing before, it was in default "step = 0 (Initial)". We have to create a step for Gravity load application.

Create a new step → ("Static, General) If we just want to do quasi-static after gravity load.

or Dynamic, Explicit → if we want to do NLTHA.

Give time
Give increments continue
Type = Auto
eg "100"

→ You can "suppress" a particular instance in Assembly by right click.

→ "Interaction type" Should be compatable with "step type".

→ To apply ground motions, we have to remove BCs and apply acceleration to those nodes.

→ make a surface at bottom to fix that. It will fix all nodes in that surface. Only nodes can also be fixed.

→ Loads : Select the step = GL
Select type eg concentrated, then node

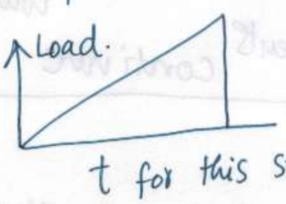
For Gravity, Component 2 = − 9810
(for self weight)

Comp 1 = X
2 = Y directions
3 = Z

→ "BC Manager" → used to remove BCs for certain steps.
or deactivate

Same with "Load Manager"
initiated → propagated etc.
step 0 step 1 - - - -

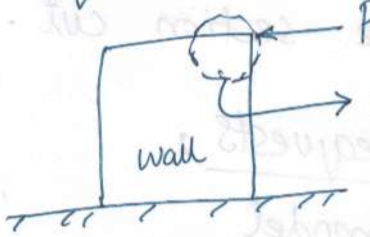
→ Load Type = Gravity
In "amplitude," we have to give its history.

Ramp ⇒ 

→ "Pre-defined Fields" → e.g used for defining pre-stressing force etc. etc.

→ "Constraints" → If you want no interaction we use constraints. e.g "Embeded region" type of constraint is used for rebars.

→ If we want quasi-static load



→ A lot of local deformation in this area.

In actual tests, we have a beam on top of wall and we push that beam. In model also, we add rigid plate to avoid local deformation.

→ To apply an imposed deformation, the option is in BC.

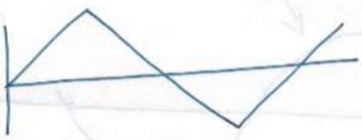
→ " " " Load " " " loads.

→ BCs → Displacement / Rotation → select the set → U1 → Uniform → Ramp

└ you can create an amplitude from here also.

→ Amplitudes → to give loading / displacement / Acc history. e.g

User Defined



Time	Amp
0	0
0.5	1
1	0
1.5	-1

→ In ABAQUS, we can do
- ✓ mass scaling
- ✓ Material "

to save time.

→ To take outputs
- Field Output Requests
- " " "

Field Output requests → whole model
"every n increments"
or " " x secs"
Make differents sets

History Output requests → e.g for top deformation.

For Create ~~Set~~ → element → continue

its just like section cut.

<u>History Output requests</u>:

Not for whole model

Set → Disp → U → U1

Shear stress = S12

→ Analysis → Jobs → Model → continue

Right click → Check data
- Monitor
- Warnings

Submit — Run

→ Results → Field Output
right click → Plot → History

because we used "smoothing" option.