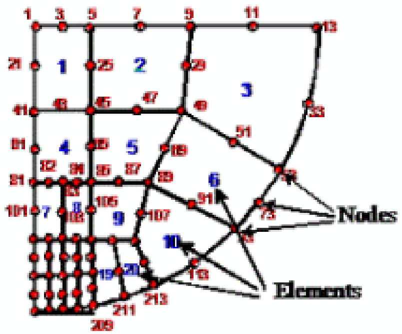
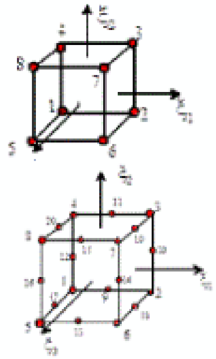
Review

Features of FE Mesh

Nodes: Used to track motion of points in solid
Elements: Main purpose is to interpolate displacements between values at nodes.

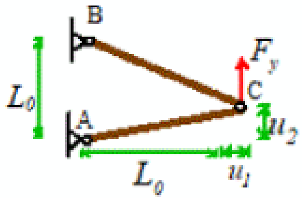
ABAQUS offers linear (nodes at corners) and quadratic (nodes at mid-sides) elements



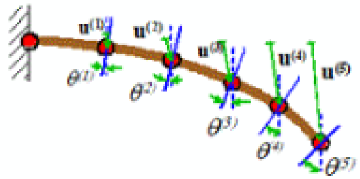


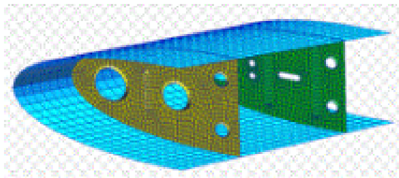
Special element types

Truss: Special displacement interpolation for 2 force members
Beam: Special displacement interpolation for slender member. Have rotation DOFS/moments
Plate/Shell: Special displacement interpolation for thin sheets that can deform out-of-plane. Rotations/moments



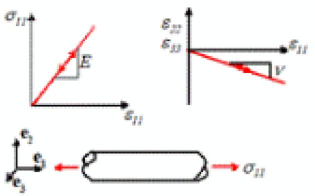
Deformed



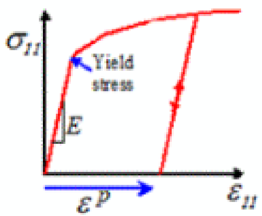


Materials (Some examples)

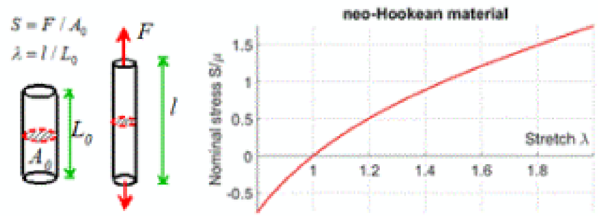
Linear Elasticity: OK for most materials subjected to small loads
Plasticity Metals beyond yield
Hyperelasticity: Large strain reversible model used for rubbers
Viscoelasticity: Time dependent material used for polymers/tissue



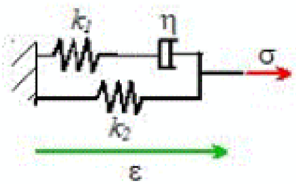
Linear Elastic



Elastic-Plastic

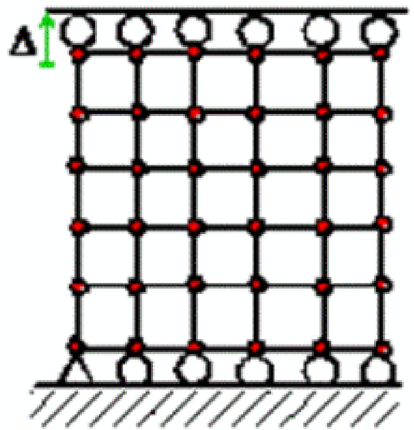


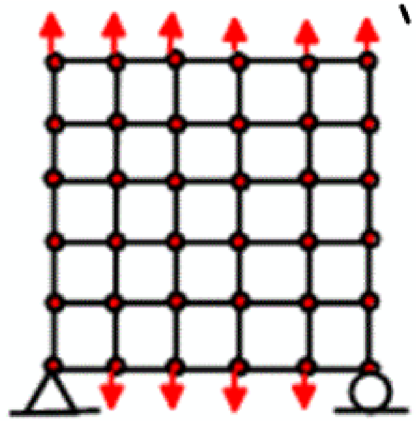
Hyperelastic



Visco-elastic

2.3 Loading





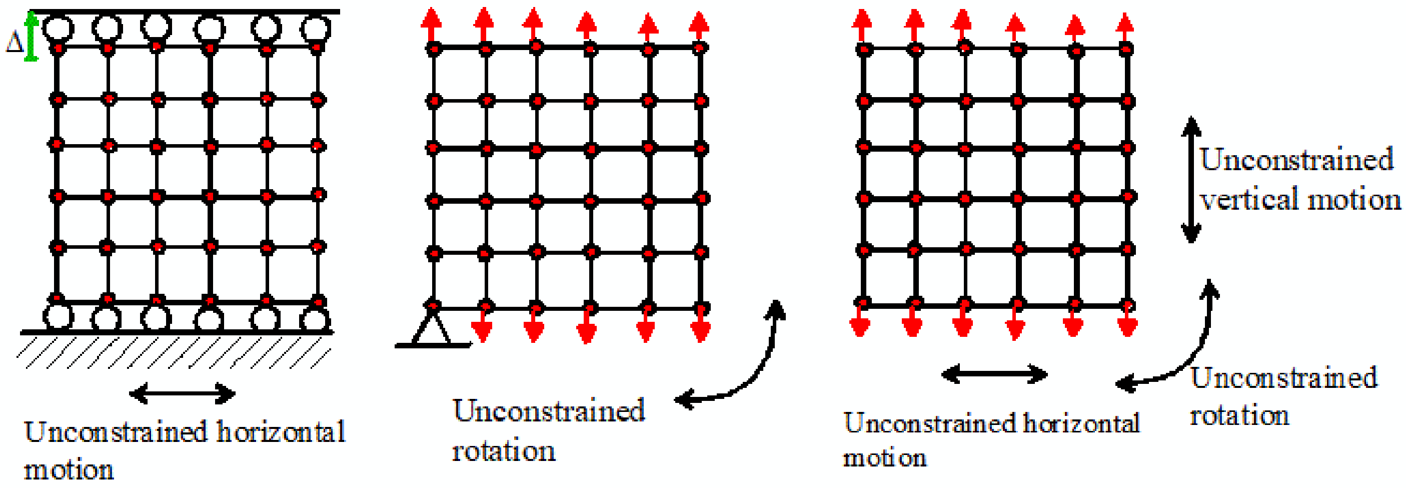
Choices:

(a) Displacements at nodes
(b) Forces on nodes
(c) Pressure on element faces
(d) Body forces inside elements

For beams / plates / shells / rigid bodies can apply rotations & moments

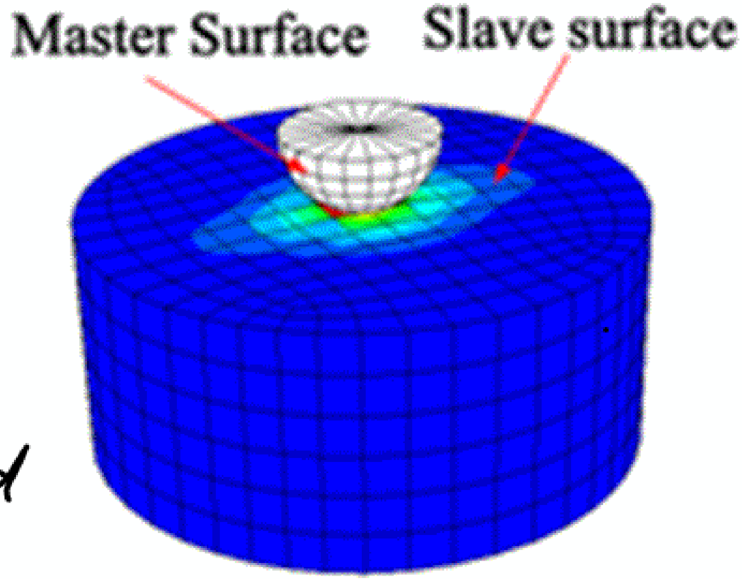
Default: Node can move freely
Element faces have no pressure

For static analysis BCs must prevent "rigid body" motion



All these will fail

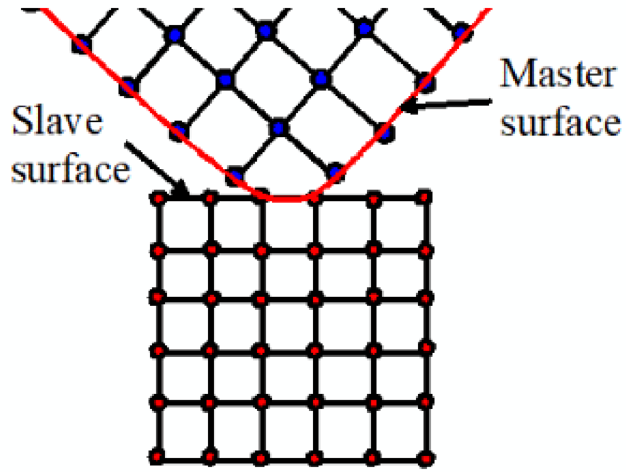
Contacts



Choices:

(1) "Node Based": nodes on "slave" surface can't penetrate "master"

(2) "Surface Based" constraint enforced in average sense



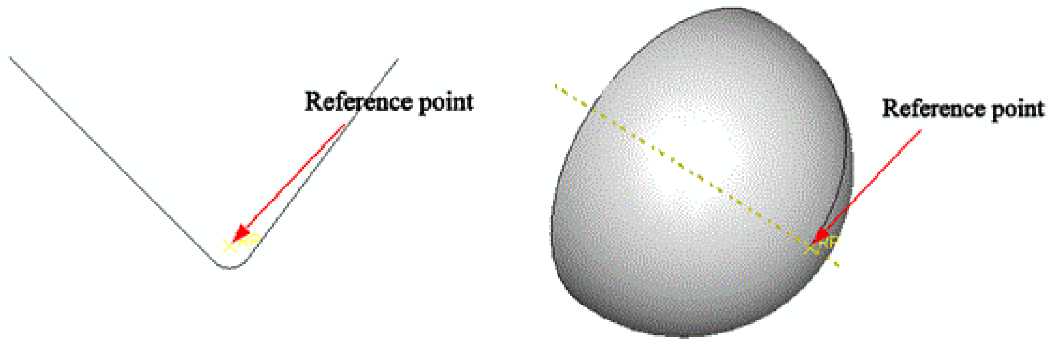
(1) more robust

(2) gives better contact pressure

Rigid surfaces (useful for contacts)

Types: Analytical: shape of surface defined by equation

"Discrete": must be meshed



Control motion with "reference node"

Contact Properties:

"Normal" behavior: "Hard": No overlap / no tension
"Soft" penetration resisted by pressure - give stiffness

"Tangential": Coulomb friction, frictionless etc

2.4 Analysis Techniques

Options (choose in "step" module)

Static ($\underline{F} = \underline{0}$)

Dynamics:

Explicit: good for impacts
robust; needs small time steps

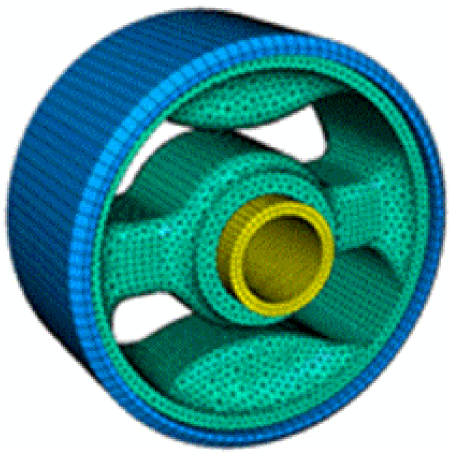
Implicit: for longer simulations

other types exist

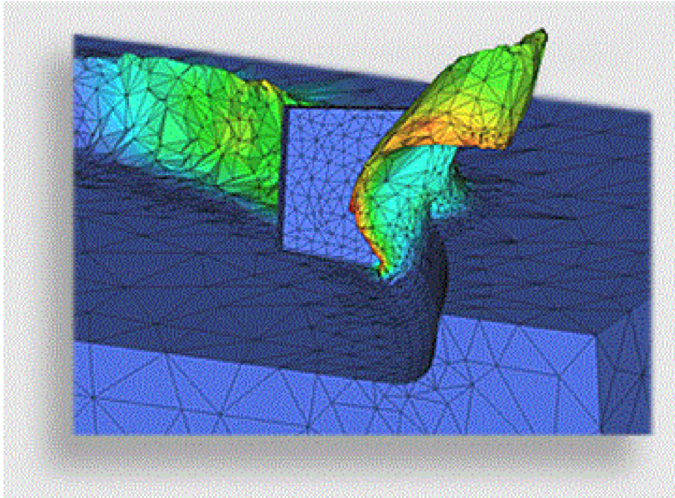
NLGEOM flag: specifies whether deformation is small or large

NLGEOM off: F = ma solved on undeformed mesh

on: F = ma on deformed mesh



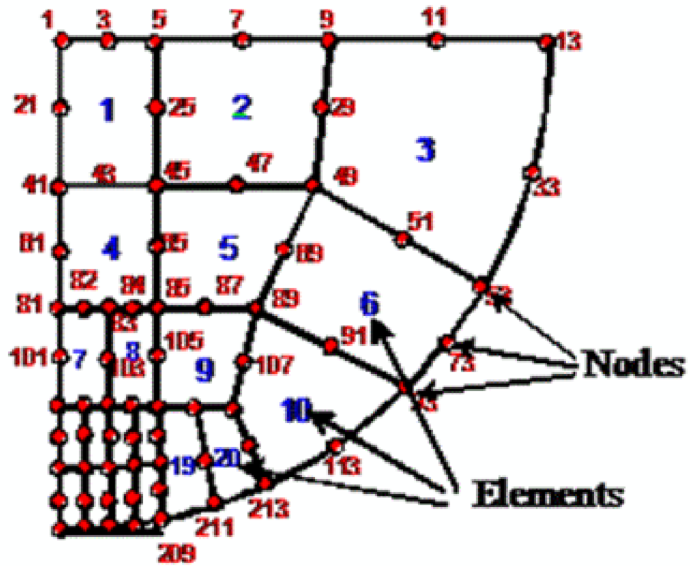
OFF



On

Need "on" for buckling.

Static Analysis



(1) NLGEOM off, elastic, no contacts
FEA solves

$$[K]\,\underline{u} = \underline{F}$$ ← forces on nodes

$\underline{u}$: vector of unknown displacements

(2) Anything else : FEA solves

$$\underline{R}^{(a)}(\underline{u}^{b}) = \underline{F}^{(a)}$$

$\underline{R}^{(a)}$: Internal force on a^{th} node

$\underline{u}^{b}$: unknown

$\underline{F}^{(a)}$: ext force

Solved using Newton-Raphson (guess & check)

N-R method may not succeed

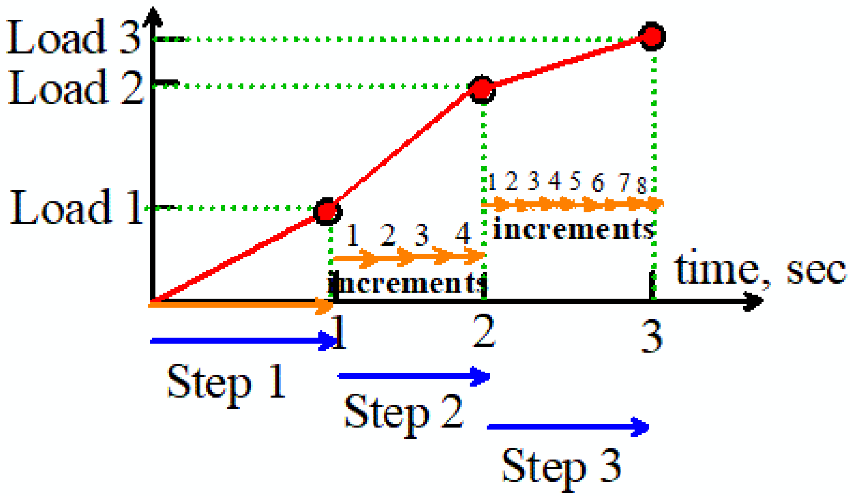
Helpful to apply load gradually

Load Steps and Increments

User applies loads in "steps"

"Steps" sub-divided into "increments"

ABAQUS automatically reduces increment size when analysis fails to converge



If timestep cut below minimum simulation aborts

Solving convergence problems (hard)

(a) Check boundary conditions

(b) Build up simulation in stages

(c) Problems typically caused by:
  (1) Contact: try different options; try different meshes
  (2) material behavior: try changing properties
  (3) Incompressibility – try "hybrid" elements

If these fail try: (1) Dynamics – increase mass to silly values.
  (2) Try "viscous stabilization" in step
    – can influence results
    – use with caution

Dynamic Simulations

FEA solves $[M]\frac{d^2\underline{u}}{dt^2} = \underline{F} - \underline{R}(\underline{u})$ (1)

Explicit: Forward Euler

March through time with small timesteps
- conditionally stable : $\Delta t <$ time for a wave to pass thro smallest element

Implicit: Backward Euler

Solves (1) with Newton-Raphson
Always stable
N-R costly.