



# Pre processor (Abaqus CAE)

Geometry

Physics of the problem

Physical properties

Boundary conditions

Meshing

Write input files



# Solver (Abaqus Standard or Explicit)

Read FE model description  
(nodes, elements,  
properties, BC)

Integrate finite element  
matrices

Assemble the global  
matrices

Solve the problem for the  
essential variables

Calculate derived  
quantities for post  
processing, write output



# Post processor (Abaqus CAE/Viewer)

Load result database & mesh

Calculate derived quantities  
(invariants, criteria, averages)

Display the results in several  
ways

Extract data (1D or 2D plot) and  
export to Text files

# What is done in the FE solver ?

## Linear statics FE solver

Assemble the mathematical problem

Solve the problem for the displacement  $u$

Calculate strains & stresses

Assemble stiffness matrix

Assemble load vector

Solve the linear system of equation

$$\varepsilon = \text{grad}(u) = B u$$
$$\sigma = C \varepsilon$$

# How is the stiffness matrix assembled?

