

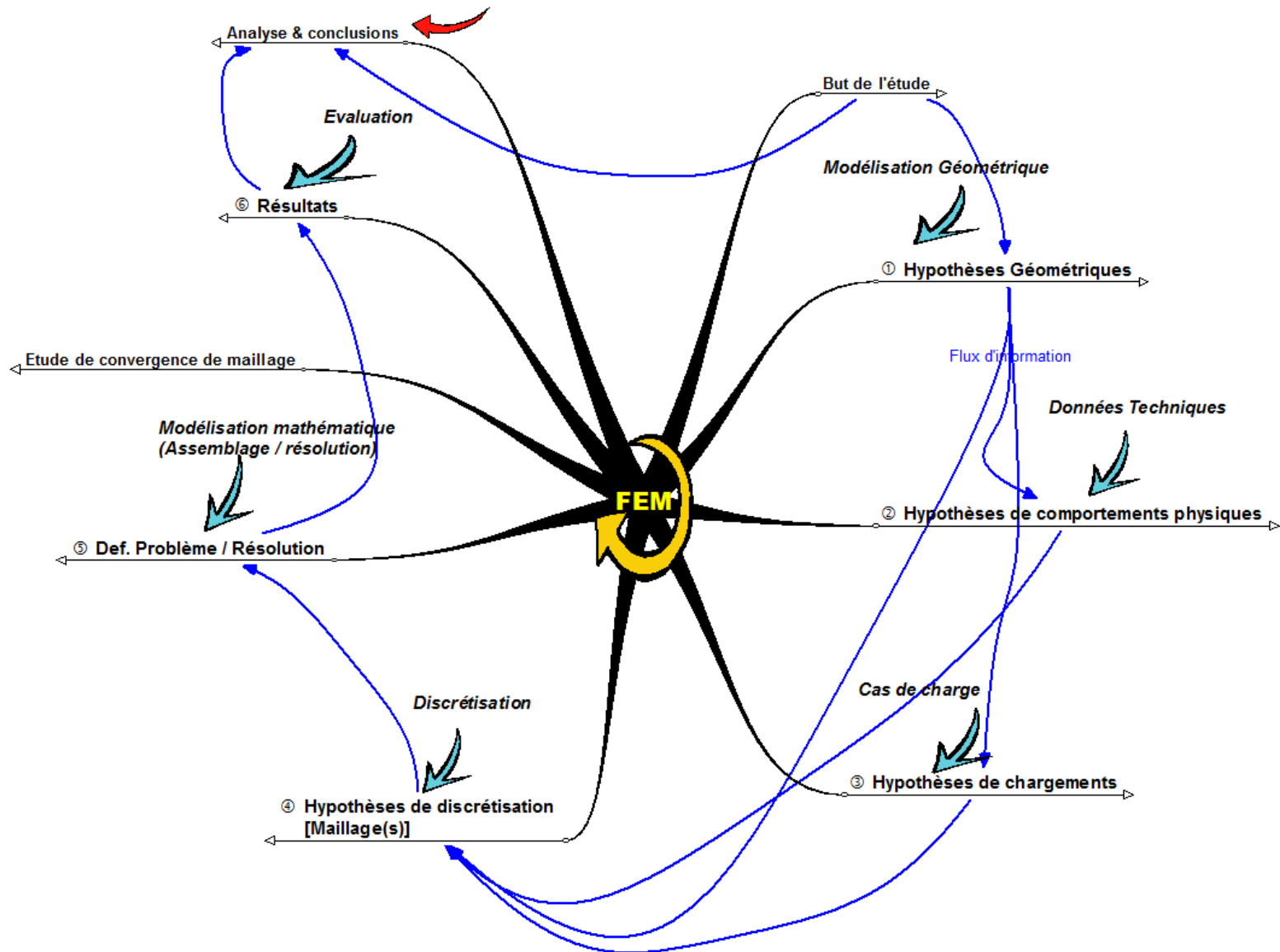
FINISH



START



But de l'étude, cahier des charges



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?

