OCCGeometry: An Implementation of the Geometry Component based on OpenCascade

- can read IGS and STEP files, but volumes have to be defined

  ToDo: Add an interface to build volumes from faces, and perhaps to modifiy the geometry (delete, add features)

  Transfer properties from CAD file

- SimpleSolidConstruction-Interface is implemented

  ToDo: an interactive user interface ?

- implements Approximation interface: delivers a polygonal approximation of the (exact) geometry

what does it do:

OCCGeometry stores a list of Volumes

When requesting an approximation, shape healing is performed:

- check if all volumes are closed
- check for intersections/overlap between all volumes, reconstruction at intersections
- inner cavities are found: a solid is built from outmost shell, from which all volumes are substracted
  -> cavities remain
- create a surface triangulation
  different surface triangulation algorithms can be chosen
- surface triangulation elements carry attributes
  “PatchID”, “LeftID”, “RightID”
- result: a “waterproof” geometry, where
  joining volumes share common faces