SALOME 2.2.0 New Features (17 February 2005)

Common information

OPEN CASCADE is pleased to announce SALOME 2.2.0. The version 2.2.0 is a SALOME maintainable release that contains major developments, bug fixes and minor improvements.

The SALOME 2.2.0 version is targeting the Redhat 8.0 platform as a reference. Please see the README file (on the installation CD) for SALOME installation and execution.

The SALOME 2.2.0 contains many new features, bug fixes and minor improvements since the last SALOME 2.1.0 (December 2004). This document presents only major and the most important new features since the SALOME 2.1.0 version. The list of bug fixes and improvements can be found in the bugtracker.

New features

- ***Update of building blocks functionality***
    - o Performance of major functions used in building by blocks was improved in several thousand times! It concerns building of mesh (propagation hypothesis), gluing of solids, building of groups
    - o Special correction class was implemented to fix problems with quasi-block solids. This class can remove internal edges which are produced by cut of spheres and cylinders, can glue also tangent edges to have only one.
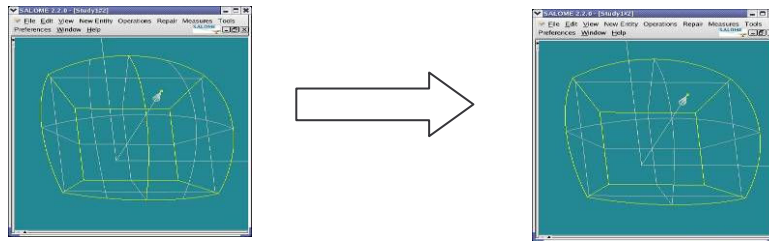

Figure 1. The result of applying of correction (removing internal seam edge)

- ***MED2.2.0 porting***
    - o SALOME now supports MED 2.2.0 as well 2.1.6 files

- ***Quality controls improvement***
    - o Aspect ratio 3D quality control is implemented
    - o Borders and multi connections quality control is introduced in this version
    - o New control - edge length control for length on links in mesh
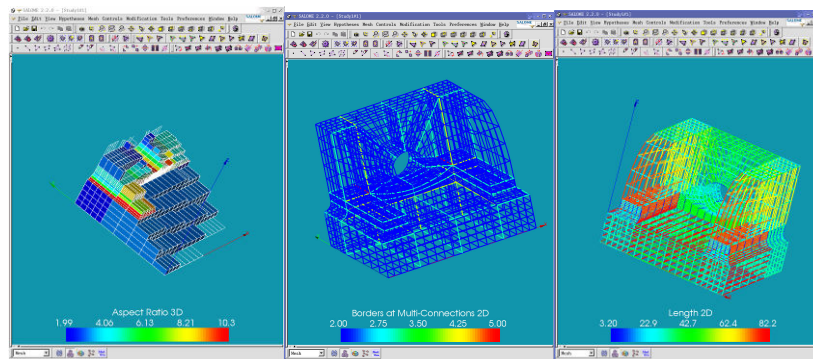   - o New presentations mode: display of edges, faces and volumes (together or in combinations)

Figure 2. New quality controls

- ***I,J,K mesher improvement***
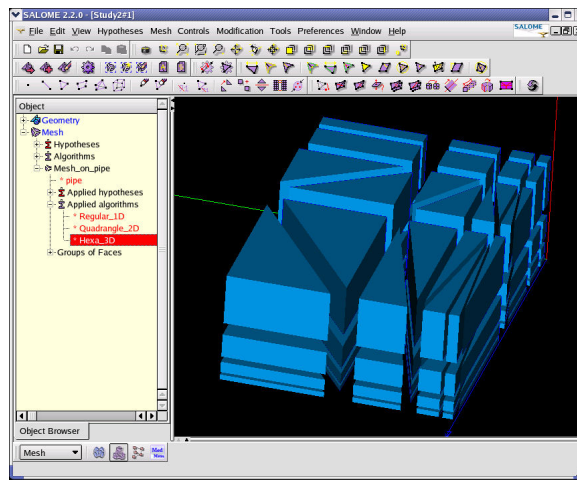   - o Now if one face of the block built with triangulation, IJK mesher can generate prismatic mesh

Figure 3. Prismatic mesh built by IJK mesher

- *Extrusion along a path.*
  - Now mesh generation becomes more advanced. User can generate "pipe" meshes using geometry curve as a basis!
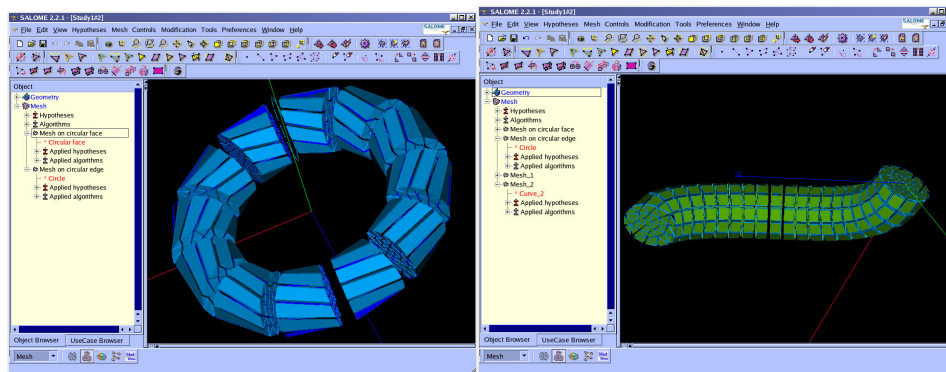

Figure 4. Pipes generated on curves.

- *Pattern mapping on mesh elements*
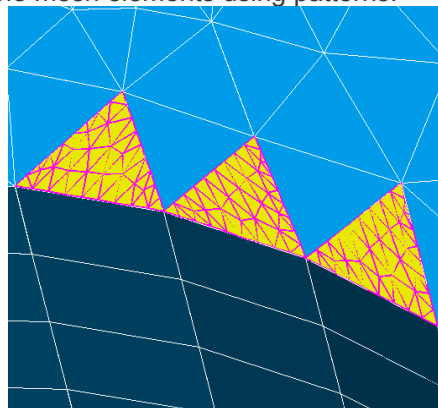 - New feature was added. Refine mesh elements using patterns.

Figure 5. Pattern mapped on three elements of the mesh

- *Improved memory management*
  - SALOMEDS server
  - Improved memory management in SMDS structures of SMESH module
  - Using of CASCADE template collections instead of STL standard one allows to optimize memory management in SMDS

Maintenance

In order to report a bug or propose an improvement, please use your account in the bug tracker (http://ocsprojects.opencascade.com/bugtracker). Please report bugs and improvements mentioning exactly your SALOME release. Please address all other questions (including bug tracker account requests) to salome-maintenance@opencascade.com.