Over the last decade, the improvements in computer hardware and software have brought significant changes in the capabilities of simulation software in the field of nuclear applications. New computer power made possible the emergence of simulations that are more realistic (complex 3D geometries being treated instead of 2D ones), more complex (multi-physics and multi-scales being taken into account) and more meaningful (with propagation of uncertainties).

Since 2001, in order to facilitate and improve this process, CEA and EDF have developed a software platform named SALOME\(^1\) that provides tools for building more complex and integrated applications. The tool is dedicated to the code environment: integration with CAD modules, meshing of CAD models, definition of input files, codes coupling and visualization. The platform has been built with collaborative development in mind and is therefore available under the LGPL license (http://www.salome-platform.org). SALOME provides modules and services that can be combined to create integrated applications that make the scientific codes easier to use and well interfaced with their environment.

SALOME is being actively developed with the support of EURIWARE/Open Cascade with 10 years of development effort of a very committed and dedicated team. SALOME is used in nuclear research and industrial studies by CEA and EDF in the fields of nuclear reactor physics, structural mechanics, thermo-hydraulics, nuclear fuel physics, material science, geology and waste management simulation, electromagnetism and radioprotection.

### GENERAL PURPOSES FOR CEA AND EDF

Many projects at CEA and EDF now use SALOME, bringing technical coherence to the software suites of these companies with the following purposes:

- Providing an integrated environment dedicated to the numerical simulation of physical phenomena.
- Responding to the specific demands for quality in the context of civil nuclear applications.
- Enabling elaborate schemes around legacy and state-of-the-art physics codes (workflows, code coupling).
- Taking advantage of high performance computing and visualization.

### MAIN FUNCTIONALITIES & TECHNICAL CHOICES

#### User interface

The platform provides an environment which covers a complete study, starting from a CAD component to define the geometry up to the visualization of the results, coupling different codes through a common data exchange model and a supervision / coupling tool.

- Two different modes of interaction with the SALOME components are systematically provided:
  - A graphic interface coupled with 3D graphic interaction (Qt4, VTK),
  - A text interface based on the Python language.

Both modes provide the same set of functionalities and SALOME offers easy short cuts from one mode to the other.

#### Component embedding and solver integration

As a platform for numerical simulation, SALOME has a very versatile and modular architecture that can be extended with additional commands or modules developed either in Python or in C++.

- It is possible to integrate codes ranging from legacy ones to state-of-the-art ones (written in Python, C++, C or Fortran).
- Component wrapper generators are available in order to facilitate the integration process.

#### Workflow supervision

With the supervisor of the platform, a user can define and control the execution of complex interconnected scientific applications on computer networks and clusters. They may be run either interactively or in batch mode.

### DEVELOPMENT FACT SHEET

- **Project kick off**: 2001.
- **Development team**: 20 persons resulting in 200 eng.years.
- **Verification**: 3 700 tests.
- **Bug tracking**: 500 corrections and evolutions / year.
- **Platform size**: 1 200 000 lines (90% C++, 10% Python).
- **Distribution**: one major version every two years, maintenance versions every six months.

### USERS

- **300** users at EDF and CEA
- **4 000** external users
- **12 000** downloads / year (web site)

**www.salome-platform.org**

---

1. Copyright © 2007-2010 CEA/DEN, EDF, Open Cascade
Copyright © 2007-2007 CAEN, EDF, Open Cascade, CAD/CCR, LIP6, CEDRAT, LEG, PRINCIPIA R&D, BUREAU VERITAS
Mesh and field management

- The platform relies on an internal data model that describes meshes and fields that are stored as sequences of HDF5 structures. Distributed meshes are taken into account.
- Interpolations are also handled in order to manage different meshes which are adapted to each simulation.

> Figures 1 - 2

DOWNLOAD SALOME

SALOME can be downloaded from the web site: http://www.salome-platform.org for several LINUX distributions and WINDOWS. The site provides tutorials, a forum section and gives access to user documentation.

> Figure 3

SERVICE AND SUPPORT

EURIWARE and Open Cascade provide a whole range of services for SALOME towards end-users including technical support and specific training.

Support services are available within an “à-la-carte” support program particularly suited for universities and academic organizations as well as for small or larger industrial companies. It provides:

- **Answer to helpdesk query** for expert help concerning a one-shot technical issue, delivered by mail or by phone within a guaranteed time frame.
- **Technical support** for complex problem solving requiring the help of a qualified engineer.
- **Expert consulting** delivered on the end-user premises by one of the SALOME expert.
- **Assistance** to create SALOME extension modules or solver integration.
- **Patch request** for an immediate access to correction, bug fixing and intermediate certified releases.

For more details, consult:
http://www.salome-platform.org/service-and-support/available-programs

Moreover, SALOME training sessions are organized on a regular basis and are available for end-users willing to familiarize themselves quickly with SALOME or reach a high level for handling complex studies.

Other training sessions on CAE solvers that are integrated with SALOME (such as Code_Aster®, Code_Saturne®), are provided by partners of the SALOME ecosystem.

Consult www.salome-platform.org to find the date of the next training session and book online.

SALOME ECO-SYSTEM

SALOME can interoperate with advanced CAD translators and commercial meshing algorithms.

**Partners**

- **CAE LINUX(r):** Engineering Linux distribution.
- **DISTENE:** provides commercial advanced and robust meshing algorithms BL-SURF and TestMesh-GHS3D.
- **CS:** development of SALOME components.
- **DATAKIT:** provides data exchange technologies.
- **DELTACAD:** Code_Aster® in SALOME.
- **INCKA:** Code_Saturne® in SALOME.
- **LOGILAB:** development of SALOME components.

SALOME 6 AND BEYOND…

The current development effort of the SALOME team encompasses the following topics:

- Parallel visualization (Paraview integration).
- Improvement of hexahedral mesh generation capabilities.
- Improved capabilities and services for the mesh and field data model (MED).
- Enhanced functionalities to access high performance computing resources.
- New integration tools for standalone codes.

ACKNOWLEDGMENT

Recent efforts in the development of SALOME for parallel computation have been supported by System@tic, Paris region system and ICT cluster, in the frame of the IOLS, EHPOC and OpenHPC projects.
The SALOME GEOMETRY module is used to define the 3D environment of a simulation study. It provides a rich set of commands to create, edit, import or modify a complex 3D model. The module is powered by a geometry kernel developed by OpenCASCADE which provides a Boundary representation of the model (BRep) and maintains the topological structure required by the subsequent meshing operations. Broadly, the geometry module is used either to create a geometrical model from scratch or to edit and prepare a model that has been designed in an external CAD modeller and then imported into SALOME.

Generic shapes such as cylinder, sphere, torus or sketch based objects can be created and located into space. Boolean operations such as "cut", "fuse", "intersect" enable shapes to be combined into more complex shapes. "Fillet" and "chamfer" operations can be applied on edges of the solids. SALOME can import geometry from IGES, STEP in BREP(r) and ACIS(r) format. It also provides a powerful set of shape-healing functionalities that can be used to simplify the model or to repair poorly defined imported models.

The GEOMETRY module functionalities can be accessed through the graphical user interface (GUI). They can also be accessed programmatically in the SALOME Python execution engine. Moreover, SALOME allows user interactions to be recorded into a command script. This function is very useful when starting to build complex automated scripts and lets one get familiar with the geometry module of SALOME.

MAIN FUNCTIONALITIES
Import of CAD models:
- Natively supported formats: ACIS, BREP, STEP, IGES
- Other formats available through commercial components, upon request: CATIA V4 / ProEngineer (c) / SolidWorks / SolidEdge / Parasolid / Nx
Creation / modification of CAD models:
- Basic objects: point, line, circle, ellipse, arc, curve, vector, plane
- Sketching: 2D sketch, 3D sketch
- Primitives: box, cylinder, sphere, cone, torus, rectangle, disk
- Topology objects: edge, wire, face, shell, solid, compound; explode object to sub-shapes
- Transformations: translation, rotation, mirroring, scaling
- Boolean operations: fuse, common, cut
- Extended operations: extrusion, revolution, chamfer, fillet, pipe
- Grouping objects
Shape-healing:
- Suppress faces, close open contour, remove internal wires, remove holes, sewing, glue faces, check free boundaries, check free faces, change orientation, add point on edge

Measures:
- Point coordinates, center of mass, inertia, bounding box, minimum distance, tolerance, angle

Export of CAD models:
- Supported formats: ACIS, BREP, STEP, IGES
- Integration of external CAD reader / writer
Visualization:
- Display / erase, change color, transparency, display mode (shading / wireframe), number of isometric lines, etc.

SALOME allows user interactions to be recorded into a command script. This function is very useful when starting to build complex automated scripts and lets one get familiar with the geometry module of SALOME.

Figure 6: CAD design of tritium unit for the HCLL Test Blanket Module in ITER (CEA/DEN)
Figure 7: Vibration behaviour of the stator of a 900MW electrical generator (EDF/R&D/AMA)
Meshing

In this module, the 3D solid shapes defined in the GEOMETRY module are transformed into finite-elements, tetrahedrons or hexahedrons. The MESHING module is used to create and edit the 1D/2D/3D mesh data and includes a variety of different open source meshing algorithms. SALOME can also be interfaced with advanced meshing algorithms developed by 3rd parties. A concept of sub-meshes can be used to take into account the specific features of the geometrical model. For every sub-mesh, a different set of meshing conditions can be applied. Mesh effective refining can be performed using pattern mapping. A complete toolbox enables the user to verify the mesh quality and to perform local modification or adjustment. Transformation operations such as translation, rotation, mirroring, extrusion, and revolution can be used to produce complex meshes or compounds. Meshes can be grouped to facilitate the visualization and help the definition of initial boundary conditions. Filters can be effectively used for group creation. Mesh data can be exported under various file formats compatible with most of the third-party numerical simulation software (i.e. MED, DAT, UNV formats).

All mesh commands are also available programmatically via the python interface, allowing scripts to handle complex studies and to simplify the management of repeatable or iterative tasks. The powerful SALOME plug-in mechanism allows additional meshing strategies to be easily integrated.

MAIN FUNCTIONALITIES

Open source meshing algorithms:
- Wire discretization
- Triangulation (Mefisto 2D algorithm)
- Quadrangle (mapping)
- Hexahedron (i-j-k algorithm)
- Tetrahedron (Netgen)
- 3D Extrusion

Commercial meshing algorithms (available upon request):
- Distene BL-SURF
- Distene TetMesh-GHS3D
- Distene Hexotic

Mesh modification:
- Add / remove nodes, elements
- Diagonal inversion
- Splitting of quadrangles to triangles; joining of triangles into quadrangles
- Transformation: translation, rotation, mirroring, sewing, merging, scaling
- Smoothing, extrusion, revolution
- Pattern mapping

Import / export mesh data:
- Supported formats: MED, UNV, DAT

Mesh groups management

Measures

Visualization:
- Display / erase meshes, sub-meshes; visualization modes: shading, wireframe, shrink; change display properties (color, lines width, shrink coefficient, transparency)

Mesh data Quality controls:
- Length of edges; area, volume; free nodes, edges, faces, boundaries; skew, taper, warping angle; 2D and 3D aspect ratio; minimum angle; etc.

> Figures 8 - 9 - 10 - 11
Post-processor module

The results of the numerical calculation generated by the CAE solvers operated by SALOME can be imported and analyzed within the SALOME POST PROCESSOR module. A wide range of functionalities can be used to visualize gradient fields and check both scalar and vector results into different presentation styles (scalar map, isometric surfaces, cut planes, cut lines, plot 3D, vectors, deformed shape, stream lines, Gauss points). Deep analysis inside hidden regions can be achieved using different clipping techniques. Progress of nonlinear or non-stationary processes can be analyzed at different speeds in the animation module. Images or videos can be easily exported and inserted into presentations or reports.

MAIN FUNCTIONALITIES
Import / export mesh data with results (fields), supported formats: MED
Import / export table data:
- Supported formats: formatted ASCII files, CSV
Graphical representation of the numerical simulations results in 3D and 2D forms:

3D visualization:
- Meshes
- Scalar fields: scalar map, ISO surfaces, cut planes, cut lines, plot 3D, Gauss points (for fields built on cells)
- Vector fields: scalar map, ISO surfaces, cut planes, cut lines, plot 3D, vectors, deformed shape, stream lines, Gauss points (for fields built on cells)

2D visualization:
- Plot of 2D curves built from table data
- Results animation along a time scale.

Mesh and field data model and associated tools

In SALOME, the MED data model for meshes and fields plays a crucial role. This MED format comes from an Open Source project in EDF R&D that is anterior to SALOME. It defines normalization for the semantics of mesh, sub-mesh and data-field representations. In addition to this normalization, the project also provides a library (MED-file), which is an HDF5 implementation of the norm. SALOME meshing and visualization modules propose import/export with MED format. Therefore, codes that use SALOME for pre- or post-processing are advised to use MED-file for input and output files.

The MED format is very generic so as to accommodate meshes corresponding to a variety of computation codes. Meshes can be structured, unstructured, contain linear or quadratic elements. Connectivities can be defined by nodal representation or descending representation. Fields can include Gauss points, lie over sub-zones of the mesh, and contain integer or floating values. MED library ensures complete compatibility with previous MED versions, making the use of different SALOME versions effortless.

TOOLS PROVIDED
On top of the MED-file library, SALOME provides a MED-memory library that enables the mounting of MED content files in memory to perform algorithms over the meshes and fields. These algorithms enable fields manipulation, Boolean operations over the mesh sub zones and interpolation between different meshes. They are valuable in the context of code coupling when the data coming from a code is most of the time not directly usable by the target code, but requires some manipulation.

Other tools are also provided with the MED-memory library:
- Parallel interpolation for computing remappings between codes lying on distributed mesh representations
- MEDSPLITTER, a tool based on METIS and SCOTCH graph libraries that creates partitioned meshes for use in parallel codes
- RENUMBER, a tool that computes cell renumbering to improve the numerical characteristics of the numerical schemes running on the meshes
- Converters for VTK, UNV SAUV mesh formats

Mesh interpolation between a grid meshed with triangles and a similar geometry meshed with quadrangles

Figure 12-13: No-load flux density distribution, result of an electromagnetic calculation (EDF/R&D/THEMIS)

Figure 14: 2D Power map in a PWR reactor corresponding to a coupled neutronics/thermal-hydraulics simulation (CEA/DEN)

Figure 15: 2D interpolation between a grid meshed with triangles and a similar geometry meshed with quadrangles

Figure 16: MEDSPLITTER split of a representative elementary volume for concrete material (context: chemical degradation, CEA/DEN)

> Figures 12 - 13 - 14
Supervision & job manager module

There is an increasing need for multidisciplinary parametric simulations in various research and engineering fields. Fluid-structure interaction and thermal coupling are two examples. The simulation tools have become very sophisticated in their own domains, so multidisciplinary simulation can be achieved by coupling the existing codes. YACS is a tool for managing multidisciplinary simulations through calculation schemes. A calculation scheme defines a chaining or a coupling of calculations (SAOLME or calculation components and Python scripts).

YACS MAIN FUNCTIONALITIES

YACS module allows building, editing and executing calculation schemes.

Main functionalities of the module are:
- Create a calculation schema
- Import/export of a schema into an xml file
- Import a catalog of calculation nodes
- Edit a schema:
  - Add/Remove a calculation node
  - Connect nodes
  - Change node information
  - Undo/Redo actions
- Representation of a schema:
  - Auto-arrange schema nodes
  - Rebuild links between nodes
  - Shrink/Expand parts of a schema
- Control the execution of a schema:
  - Execute a schema
  - Suspend/Resume execution
  - Step-by-step execution and breakpoints
  - Save/Restore execution state
- Different kinds of calculation nodes:
  - Service nodes (distributed services)
  - Python nodes (inline or distributed)
  - Sequential loop node (for, while)
  - Parallel loop (for parametric studies)
  - Switch node (switch, case)
  - Optimizer loop (for optimization algorithms)

HOW TO BUILD SALOME COMPONENTS THAT CAN BE COUPLED WITH YACS

To couple calculation codes with YACS, it is essential to transform them into SALOME components. This operation requires a good knowledge of SALOME principles but in most cases, it is possible to use helper tools.

- The hxx2salome tool automates this operation to a large extent for C++ calculation codes, starting from the include file.
- The YACSGEN tool automatically generates the necessary SALOME embedment starting from a description of the selected coupling interface (for Fortran, C++ and Python calculation codes that use CALCULUM type coupling).

JOBMANAGER MODULE

DESCRIPTION

JOBMANAGER module allows creating, launching and following calculation jobs on different types of computers.

JOBMANAGER MAIN FUNCTIONALITIES

The JOBMANAGER module allows to define three types of jobs:
- User scripts.
- Python scripts launched in a SALOME session.
- YACS schemas.

The module can use different types of computers:
- Interactive computers (sh, ssh).
- Clusters managed by batch systems like PBS, LSF or SGE.

Figures 18 - 19 - 20 - 21 - 22 - 23

Figures 20 - 23: Thermal model for the study of High Level Waste (HLW) geological disposal (EDF/R&D/SINETICS).