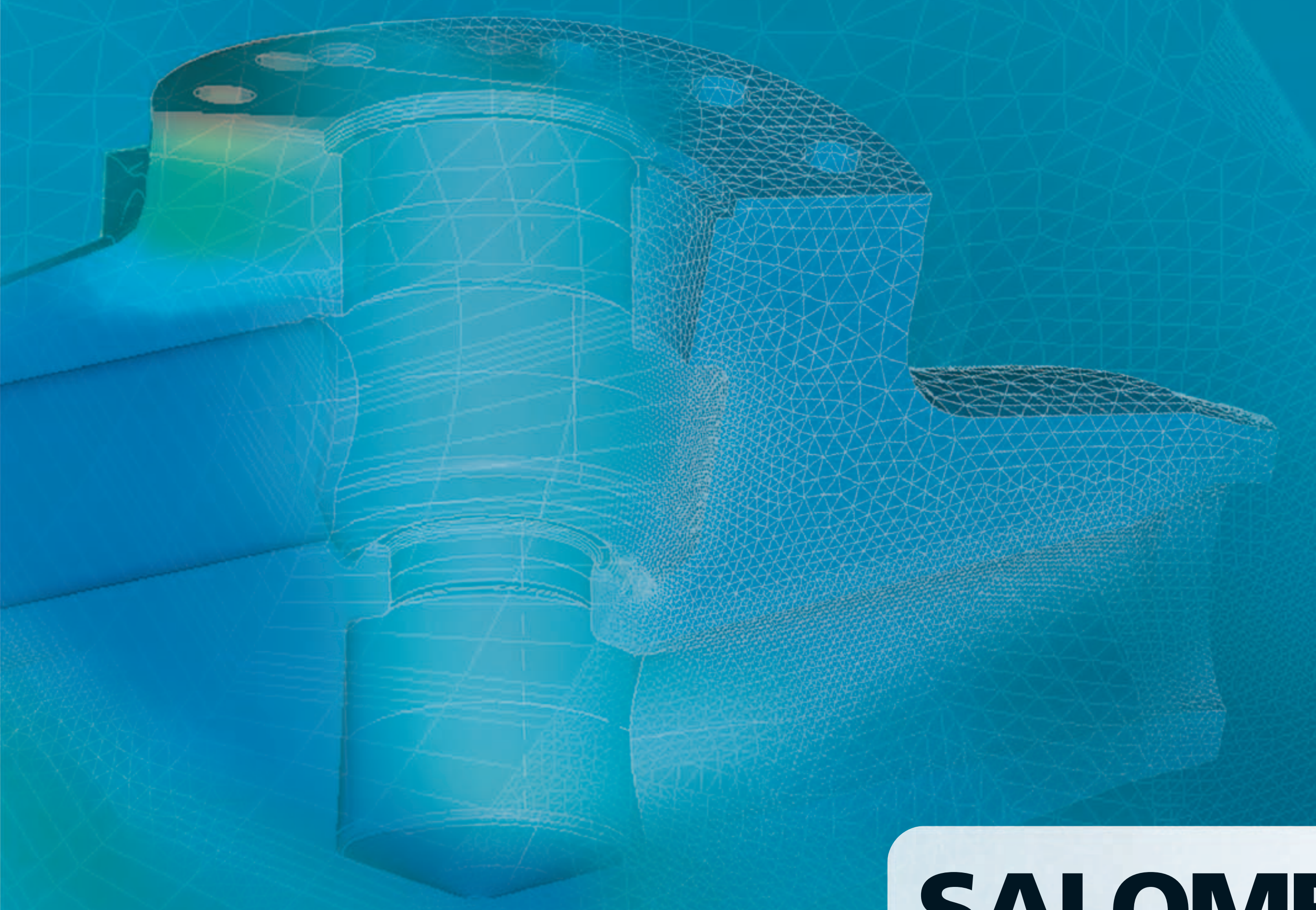
salome-platform.org



SALOME6

THE OPEN SOURCE INTEGRATION PLATFORM FOR NUMERICAL SIMULATION

cea
energie atomique • energies alternatives

EDF

OPENCASCADE

SALOME 6

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 using a collaborative development approach 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.

DEVELOPMENT FACT SHEET

Project kick off	2001.
Development team	20 persons resulting in 200 eng.years.
Verification	4 000 tests.
Bug tracking	500 corrections and evolutions / year.
Platform size	1 300 000 lines (90% C++, 10% Python).
Distribution	one major version every two years, maintenance versions every six months.

USERS

400 users at EDF and CEA

5 000 external users

13 000 downloads / year (web site)

www.salome-platform.org

SALOME KEY FEATURES

- Drives scientific software towards standardized approaches (data exchange models, technical choices).
- Supports interoperability between CAD modeling and computation software (CAD-CAE link).
- Provides a module structure that allows integration and hosting of new scientific codes.
- Facilitates coupling between computation codes.
- Provides a generic, user-friendly and efficient user interface, which helps to reduce the costs and delays of carrying out the studies.
- Is qualified by CEA and EDF for industrial and R&D studies.
- Is based on an Open Source strategy to facilitate collaborative development and to build specialized applications.

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 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.

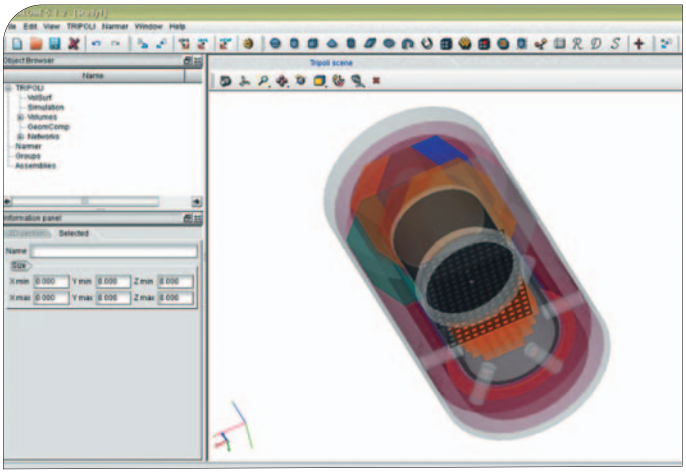


Figure 1: SALOME-TRIPOLI is a CEA application dedicated to the pre-processing of the TRIPOLI Monte Carlo code.

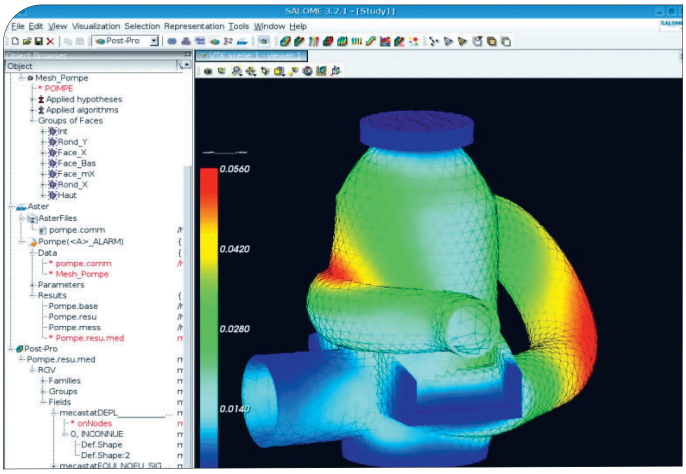


Figure 2: SALOME-MECA is an EDF application dedicated to the pre- and post-processing for the Code_Aster® structural mechanics code (http://www.code-aster.org).

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 also 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 professional end-users including technical support and specific training.

Support services are available within a "à-la-carte" support program particularly suited for universities and academic organizations as well as for small or larger industrial companies:

- *Helpdesk support* for expert needs concerning a one-shot technical issue, delivered by mail or by phone within a guaranteed time frame.
- *Technical support* for complex problem solving that requires 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.

Figure 3: http://www.salome-platform.org

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 TetMesh-GHS3D.
- *CS*: development of SALOME components.
- *DATAKIT*: provides data exchange technologies.
- *DELTACAD*: Code_Aster®[2] in SALOME.
- *INCKA*: Code_Saturne®[3] in SALOME.
- *LOGILAB*: development of SALOME components.

SALOME 7 AND BEYOND…

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

- Improvement of hexahedral mesh generation capabilities.
- Enhanced functionalities to access high performance computing resources.
- Graphical user interface to give access to high level mesh and field algorithms.
- Standardization of study data management.

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, OpenHPC, ILMAB and OASIS projects.

2 Code_Aster® is EDF's numerical simulation software for structural analysis.
3 Code_Saturne® is EDF's general purpose Computational Fluid Dynamics (CFD) software.

Geometry module

This module provides a rich set of commands to create, edit, import or modify a complex CAD model.

The module is powered by a geometry kernel based on the Open CASCADE Technology which provides a Boundary representation of the model (BRep) and maintains the topological structure required by the subsequent meshing operations.

SALOME can import geometry from IGES, STEP, in BREP(r) and ACIS(r) format[4]. 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 that allows building complex automated scripts.

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.

> **Figures 4 - 5 - 6 - 7**

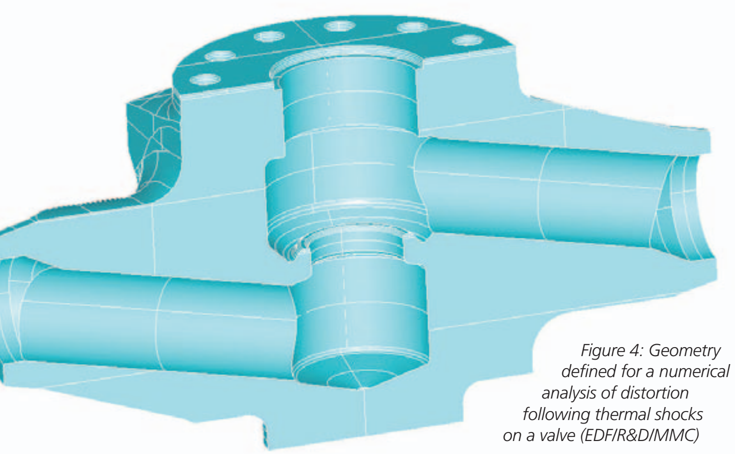


Figure 4: Geometry defined for a numerical analysis of distortion following thermal shocks on a valve (EDF/R&D/MMC)

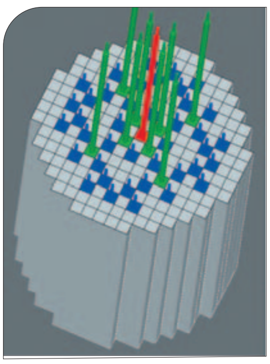


Figure 5: GEOM CAD illustrating control rod positioning. This geometry is part of a SALOME data preprocessor for the DYN3D code built in the frame of the NURESIM European project (courtesy of FZR, Forschung Zentrum Dresden)

4. Note: Additional direct CAD translators for popular CAD formats such as CATIA V4®, CATIA V5®, Parasolid®, SolidWorks®, SolidEdge®, Siemens NX®, ProEngineer® can be purchased from OpenCascade.
ACIS®, ProEngineer®, Catia®, SolidWorks®, NX®, Parasolid® are registered trademarks of their respective owners.

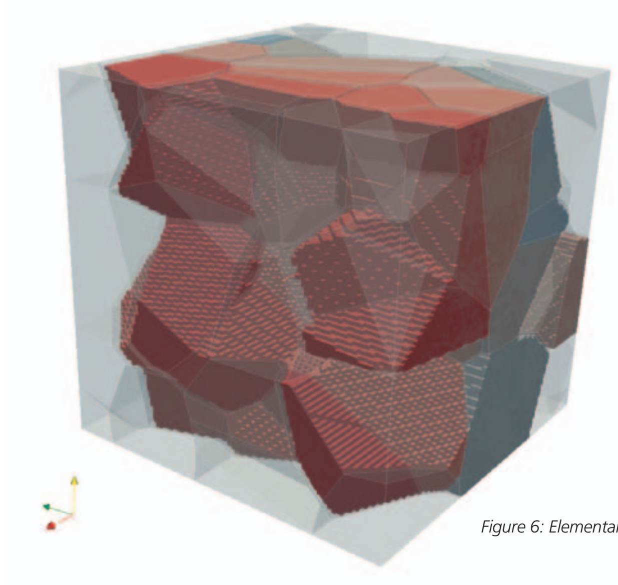


Figure 6: Elementary volume voxelization (CEA DEN)

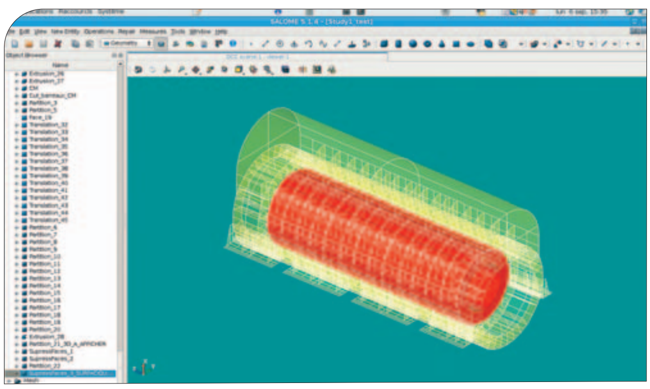


Figure 7: Vibration behaviour of the stator of a 900MW electrical generator (EDF/R&D/AMA)

Meshing module

This module transforms the 3D solid shapes defined in the GEOMETRY module into finite-elements. The MESHING module is used to create and edit the mesh data and includes a variety of different open source or 3rd parties meshing algorithms.

A concept of sub-meshes can be used to take into account the specific features of the geometrical model. A different set of conditions can be applied to each sub-mesh.

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 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.

All mesh commands are also available programmatically via the python interface, allowing scripts to handle complex studies.

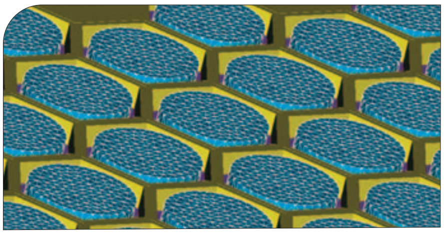


Figure 8: Visualization of the mesh of a Gas Fast Reactor fuel plate (CEA/DEN)

MAIN FUNCTIONALITIES

Meshing algorithms:

- Open Source (Wire discretization, Triangulation, Quadrangle, Hexahedron, Tetrahedron, 3D Extrusion)
- Commercial (available upon request): Distene (BL-SURF, TetMesh-GHS3D, 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
- Diagonal inversion

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

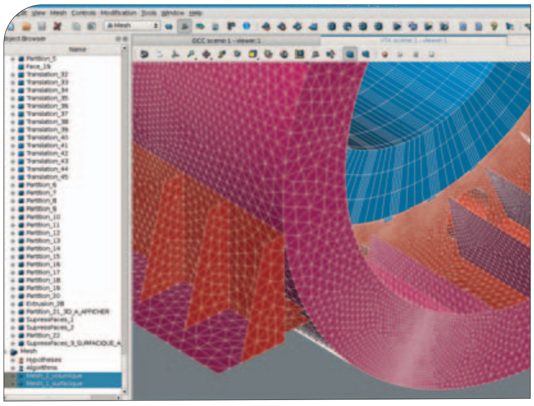


Figure 9: Vibration behaviour of the stator of a 900MW electrical generator (EDF/R&D/AMA)

Mesh Adaptation module

To improve the quality of the results of the simulation, mesh adaptation offers an effective compromise, combining a fine mesh with a low computational cost. The HOMARD®[5] module allows refinement and coarsening techniques to adapt the mesh, according to the numerical error of the simulation.

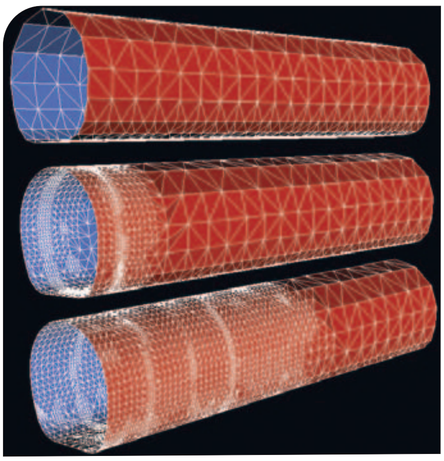


Figure 10: Mesh adaptation during the simulation of a tunnel excavation (EDF R&D / SINETICS)

HOMARD® is designed to operate in association with 2D/3D element such as triangles, quadrangles, tetrahedrons and/ or hexahedrons. The whole mesh can be conformal or not.

The selection of the elements to refine is made either by the value of a field over the elements and a threshold, by a group or by a geometrical zone. Splitting their edges in 2 refines these elements. The transition between different refinement zones is treated with special elements.

The fields can be interpolated from the old mesh to the new one.

If the boundary of the mesh is curved, the new nodes can be moved onto that line or surface.

Mesh data are imported and exported under the MED format. They are included in the tree of the Meshing module.

All HOMARD® instructions can be provided either through the graphical user interface or via the python interface.

MAIN FUNCTIONALITIES

Mesh:

- Edges, triangles, quadrangles, tetrahedrons and/or hexahedrons
- Conformal or not
- Degree 1 or 2 (exclusive)
- Groups of elements are preserved
- Equivalence of elements are preserved

Governing parameters:

- Uniform refinement
- A field: comparison of a refinement threshold and the local norm of the field or the jump between two elements
- Geometrical zone: sphere, box, cylinder
- Filtering with group of elements

Curved boundaries:

- Discrete description for lines with a specific 1D mesh
- Analytical description for surfaces: cylinder, sphere

Field management:

- Interpolation P0, P1, P2, iso-P2

Import / export mesh data:

- Supported format: MED

> Figure 10

5 HOMARD® is EDF's software for the mesh adaptation.

Post-processor module

Numerical solvers operated by SALOME generate results that can be analyzed within the ParaViS module.

This module has been improved by integrating ParaView into SALOME, and exposes all the functionalities of this award-winning post-processor tool.

A wide range of representations are available to the physicists to explore the datasets: surface, volume, gauss points... The data can then be analyzed by using one of many filters to extract significant data: clip, threshold, iso-surface, stream lines, elevation surface…

Quantitative information can be extracted using the data analysis tools: taking a selection of the data, histograms, 2D plots of time-plots are one click away.

All these features can be animated within the module to analyze time-varying data, sweep a cutting plane through the dataset, or animate a modal analysis.

This module is fully scriptable in python to create visualizations in batch when necessary or to repeat analysis on ensemble runs, and can use visualization clusters to interactively analyze large datasets.

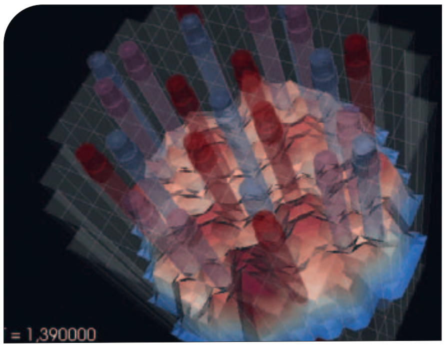


Figure 11: Power density in a coupled thermalhydraulics / neutronics calculation (CEA DEN)

MAIN FUNCTIONALITIES

Import / export mesh data with results, supported formats: MED and all ParaView-supported formats

Import / export table data:

- Supported formats: CSV

Iterative filtering of the simulation results:

- Threshold, Clip, subset selection, cutting plane

Graphical representation:

- 3D views: Meshes, scalar map, iso-surfaces, Gauss points, vectors, stream lines, …
- 2D views: 2d plots, histogram
- Spreadsheet
- Comparative views

Animation:

- Over time
- Any visualisation parameter

Fully scriptable

Remote and parallel visualization capabilities

> Figures 11 - 12

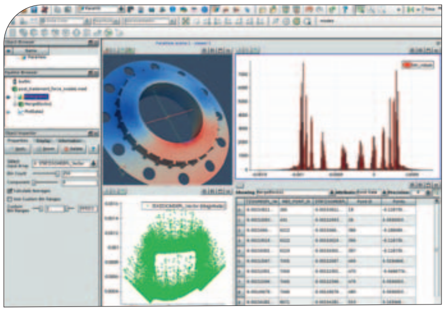


Figure 12: Postprocessing of a stress analysis study (EDF R&D/SINETICS)

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 visualisation 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.

MED format is generic and flexible enough to accommodate meshes with 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. Multi time steps fields can include Gauss points, lie over sub-zones of a moving mesh over time. 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 MED module that proposes a set of down-top sorted libraries that offers a set of services going from elementary operations on cells up to advanced operations on multi timesteps fields and meshes. The proposed set of libraries gives developers flexibility between the services and the number of prerequisites. Typical MED module use is the mounting of MED content files in memory to perform manipulations over the meshes and fields and to write them back. These algorithms enable fields manipulation with HPC constraints, 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:

- ParaMEDMEM, parallel interpolation for computing remappings between codes lying on distributed mesh representations
- CALCULATOR, to manipulate multi time steps fields contained coming from MED File easily.
- 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

> Figures 13 - 14

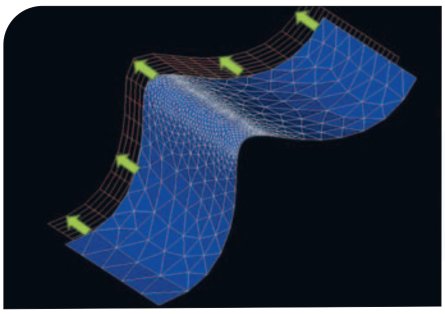


Figure 13: 2D interpolation between a grid meshed with triangles and a similar geometry meshed with quadrangles

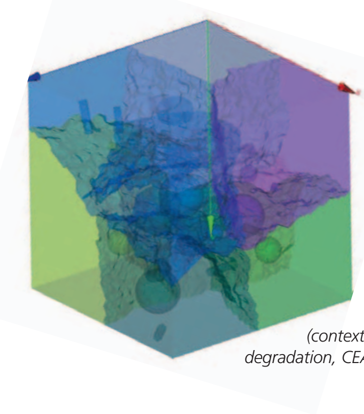


Figure 14: MEDSPLITTER split of a representative elementary volume for concrete material (context: chemical degradation, CEA/DEN)

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 tools used in numerical simulation have become very sophisticated in their own domains, so multidisciplinary simulation can be achieved by coupling the existing codes.

> Figure 15

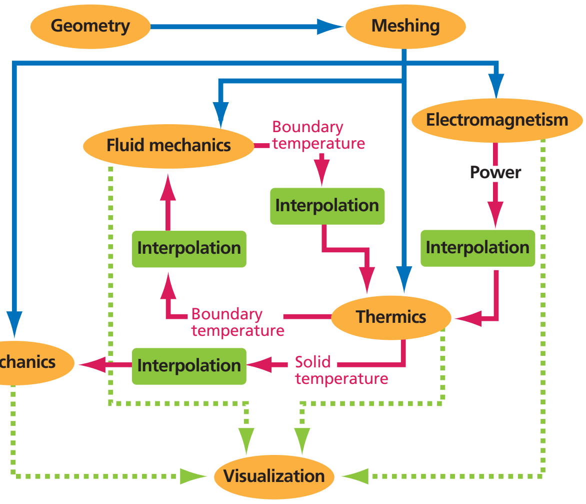


Figure 15: Multidisciplinary simulation example

YACS is a tool for managing multidisciplinary simulations through calculation schemes.
A calculation scheme defines a chaining or a coupling of calculations (SALOME 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
- Represent and visualise 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
- Use 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)

Figures 18-21: Thermal model for the study of High Level Waste (HLW) geological disposal (EDF/R&D/SINETICS).

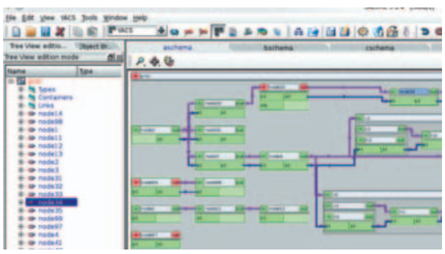


Figure 16: View of YACS GUI

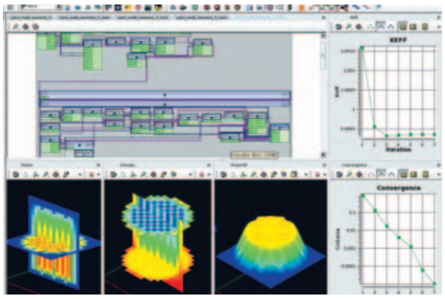


Figure 17: Execution graph of a coupled neutonics-thermalhydraulics calculation. On top of the execution graph, convergence history and 2D maps are displayed during the calculation (CEA/DEN)

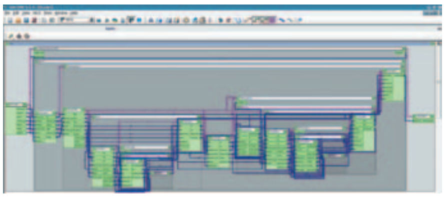


Figure 18: Calculation scheme for the repository size optimisation of several different HLW packages with the EDF thermal Code_Syrthes®, distributed on a cluster computer (EDF/R&D/SINETICS)

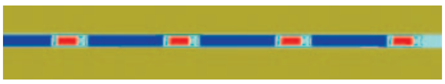


Figure 19: Geometric representation of a HLW disposal cell (EDF/R&D/SINETICS)

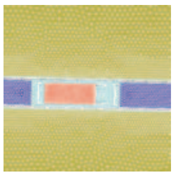


Figure 20: Zoom on the mesh of the disposal cell (EDF/R&D/SINETICS)

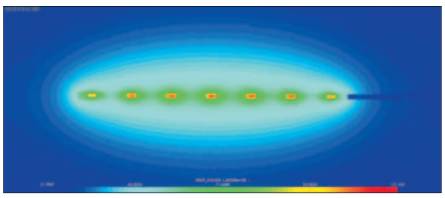


Figure 21: Temperature field after several years in the disposal cell (EDF/R&D/SINETICS)

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 YACSGEN tool automatically generates the necessary SALOME embedment starting from a description of the selected coupling interface (for Fortran, C++ and Python calculation codes).

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 (rsh, ssh).
- Clusters managed by batch systems like PBS, LSF, SGE, LOADLEVELER or SLURM.

> Figure 22

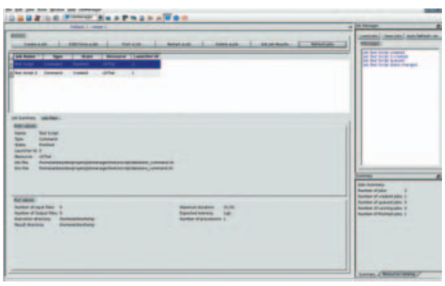


Figure 22: Example of job launching with the JOBMANAGER

CONTACT US:

Vincent Bergeaud	vincent.bergeaud@cea.fr
Vincent Lefebvre	vincent.lefebvre@edf.fr
Étienne Rossignon	etienne.rossignon@euriware.fr

energie atomique • energies alternatives

Commissariat à l'Energie Atomique
et aux Energies Alternatives,
CEA-Saclay, DEN, DM2S,
91191 Gif-sur-Yvette Cedex, France
www.cea.fr

EDF R&D
1, avenue du Général de Gaulle,
92141 Clamart Cedex, France
www.edf.com

Open Cascade
Filiale d'EURIWARE, groupe AREVA,
1 place des frères Montgolfier
78044 Guyancourt, France
www.opencascade.com

EDF RECHERCHE ET DÉVELOPPEMENT - Novembre 2011 - Le Groupe EDF est certifié ISO 14001 - SA au capital de 924 433 331 euros - 552 081 317 R.C.S. Paris - D & COM