

# SALOME7

THE OPEN SOURCE  
INTEGRATION  
PLATFORM  
FOR NUMERICAL  
SIMULATION



# SALOME7 PLATFORM



Figure 1: Screenshot of the public web site of SALOME. [www.salome-platform.org](http://www.salome-platform.org)

The SALOME software platform is an open framework that can integrate scientific solvers for modelling various physical domains and realize a wide range of computing simulations. In the context of CEA and EDF, the physical systems are typically the industrial equipments of the nuclear production plants, with as primary concern the design of new generation reactors, the nuclear fuel management, the reliability and safety of the installations, or the equipments ageing for an optimized life cycle management. The challenges are to analyse the behaviour of the systems with sufficient accuracy and performance to aid the decision making on industrial questions. SALOME comes natively with CAD modelling, mesh algorithms, 3D visualization and advanced functions for simulation data processing.

## KEY FEATURES

The numerical analysis of complex industrial systems demands from scientists and engineers to integrate most domains of physics like material science, solid mechanics and structural dynamics, fluids physics and thermohydraulics, nuclear physics and radiations, electromagnetism. These domains can be federated in one single simulation environment, the SALOME platform.

The main features of the SALOME platform are :

- **The design of the geometry representation** of physical systems (CAD modelling) and its associated discretized model (meshing functions for finite elements or finite volumes solvers).
- **The ability to integrate domain specific solvers** into normalized software components with standard interfaces to facilitate the coupling of different physical domains.
- **The supervision of computation workflows** defined as graphs of distributed software components, including CAD modelling, domain specific solvers and data processing components.
- **The analysis of simulation data**, in particular using visualization of physical fields resulting from computation workflows.

In this context, a key point of the platform is the usage of standard data models to describe physical concepts for numerical analysis, and to ensure interoperability between software components. The MED data model for example is used to describe geometric meshes and physical fields.

## MODEL OF DEVELOPMENT

The SALOME platform is actively developed by CEA and EDF, two of the key actors in the domain of energy in France, and with the support of the EURIWARE/Open Cascade company, a leader in the development of software programs for scientific modelling and computing simulation. This 12 years old partnership provides the SALOME development project with a very committed and dedicated team in scientific computing, in particular for specific applications in nuclear industry.

Moreover, the SALOME platform development relies on the integration of cutting edge third party software programs, for example the commercial advanced and robust meshing programs MeshGems-CADSurf (formerly known as BLSURF) and MeshGems-Tetra (formerly known as GHS3D) (by the Distene Company) and the PARAVIEW software including the VTK 3D visualisation toolkit (by the Kitware Company).

Recent efforts in the development of SALOME for parallel computation have been supported by System@tic, Paris region system and ICT cluster. This includes for example the projects IOLS, EHPOC, OpenHPC, ILMAB and OASIS.

## DOWNLOAD THE SALOME PLATFORM

The SALOME platform 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 and source code under LGPL licence.

## 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>

## DEVELOPMENT FACT SHEET

Project kick off	2001
Development team	20 persons
Licence	LGPL
Distribution	one major version every 2 years, and maintenance versions every 6 months.
Bugtracking	500 fixes and improvements / year.
Verification & Validation	4 000 tests.
Users	300 users at EDF and CEA, 4 000 users over the world, 40 000 downloads in year 2012

# DEVELOPING DOMAIN SPECIFIC APPLICATIONS

The SALOME platform provides a software suite to integrate domain specific tools and realize dedicated solutions for simulation of physical systems. For example, you can download from <http://www.code-aster.org> the SALOME-MECA application that is the EDF simulation environment to investigate solid and structural mechanics questions about equipments ageing, reliability and safety of production plants. This application is a software configuration that integrate the SALOME platform with the Code\_Aster solver and its computing environment.

A second example is PANTHEREV2, a numerical simulation application dedicated to industrial radiation protection. The screenshot below illustrates the computation of the average dose rate in a nuclear power plant (Figure 2).

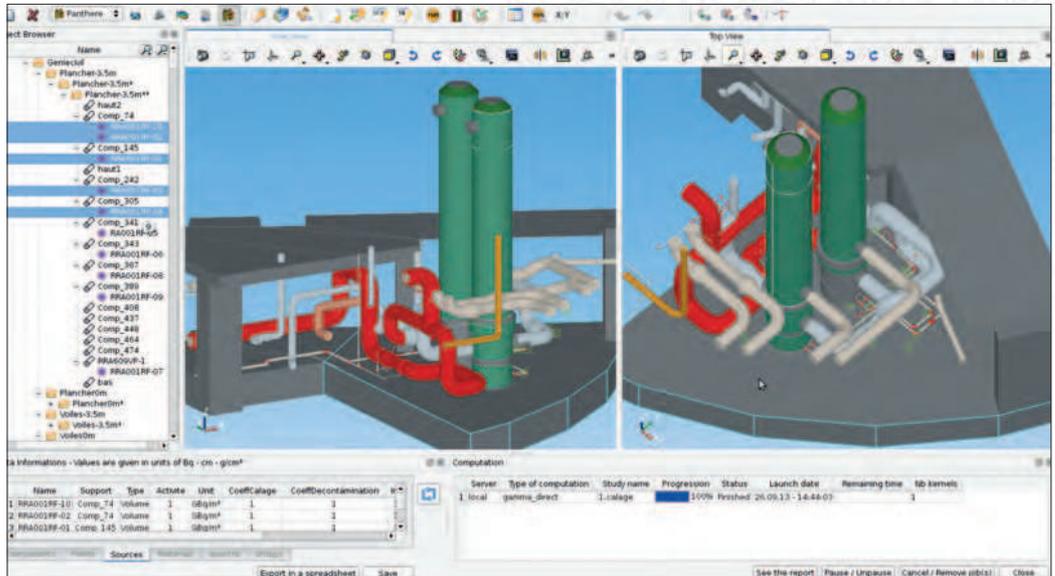


Figure2: Evaluation of the average dose rate of radiation using the PANTHERE application (EDF/SEPTEN).

The following third example is a GUI interface for GENEP+ code named PPGP and dedicated to the analysis of steam generators. It helps the user to describe the generators and its associated physical properties, and then manage the computing workflow from the meshing process to the data analysis and visualisation (Figure 3).

In this context, the SALOME platform is distributed with a Quality Assurance industrial process, including a Verification and Validation activity. The development team provides first level services and support to investigate the development of dedicated applications by third party projects.

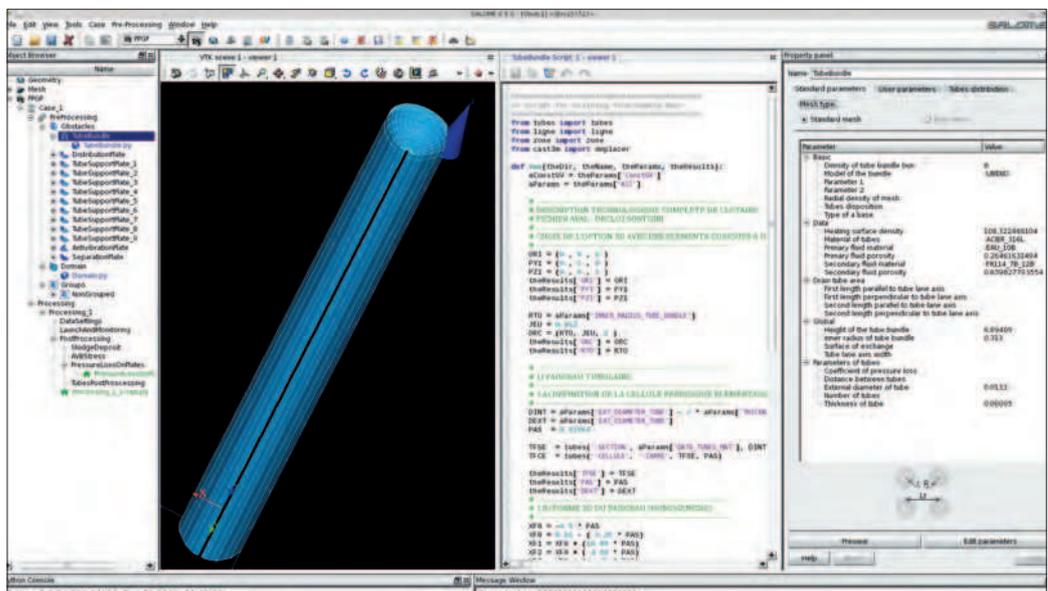
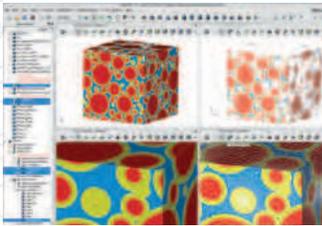


Figure3: PPGP application, at left: tree of a all elements of a steam generator, at middle: python access to specific changes, at right: dialog box to edit values of a part of the steam generator

# MODELLING PHYSICAL SYSTEMS



The SALOME platform provides you with a whole set of modules to create complex geometric models. This includes high level meshing functionalities to prepare numerical models that fit the solver requirements and simulation objectives, in terms of accuracy and performance.

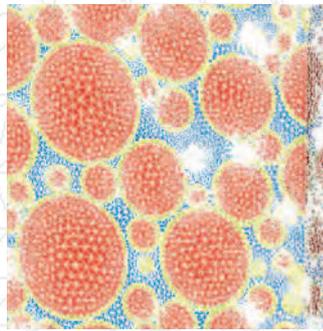


Figure 5: Representative Volume Elements generation for material studies (CEA/DEN).

## MODELLING THE GEOMETRY

The Geometry module (GEOM) provides a rich set of commands to create, edit, import or modify a complex CAD model. The geometric shapes may be designed interactively using the Graphical User Interface (GUI) or the Text User Interface (TUI) through python scripts. Each functionality of GEOM is available in TUI with facilities to work back and forth with the graphical interface. This allows to build complex shapes or several configurations of a shape with different values of its parameters.

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 exchange geometry with other CAD systems using formats as IGES, STEP, and BREP (the Open CASCADE native format), (Figures 4 & 5).

## MESHING THE GEOMETRY

The SALOME platform comes natively with a set of modules to create Finite-Elements representations of the geometry of your system (e.g. **SMESH module**) and to refine parts of the mesh representation considering the solver specific requirements and the simulation objectives (e.g. **HOMARD module**).

### The Meshing module

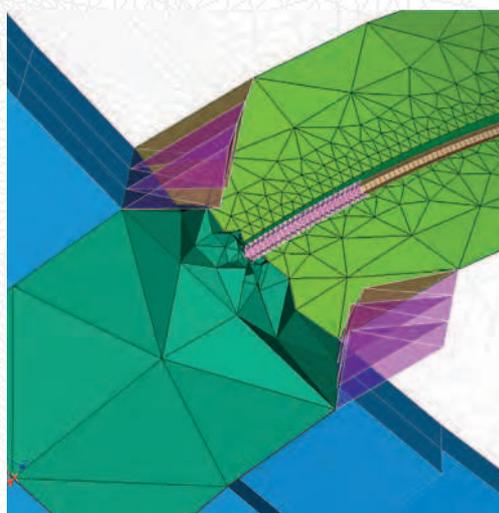
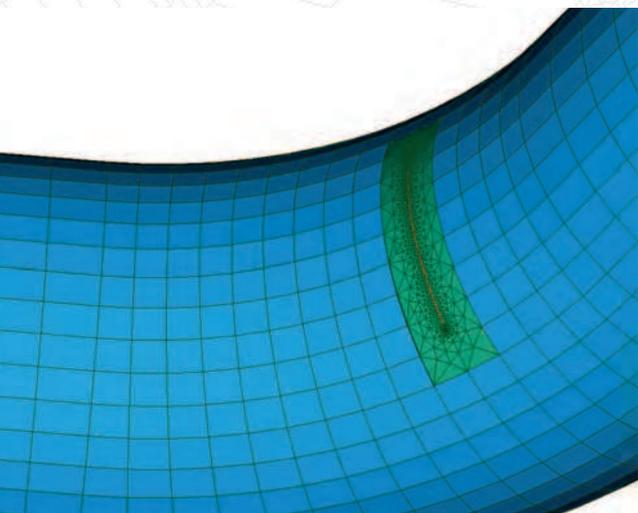
The Meshing module (SMESH) transforms the solid shapes defined or imported in the Geometry module into finite-elements. The Meshing module is used to create and edit the mesh data using a variety of different open source or 3rd parties meshing algorithms.

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, for example to mesh with smaller elements some area of the shape, or to mesh an area with another kind of elements.

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.

Mesh elements can be grouped to facilitate the visualization and help the definition of initial boundary conditions. These groups can be automatically deduced from the corresponding geometric groups, or generated using grouping criteria (named "filters" in SALOME).

As for the CAD process, the meshing process can be entirely described into python scripts (The Text User Interface of SALOME) to handle complex studies or to ensure full reproducibility and parametrization of the simulation workflow. (Figures 6 & 7)



Figures 6&7: Crack modelling for ageing analysis of a piping system. Left: external view of the complete crack. Right: internal view of the detailed meshing strategy

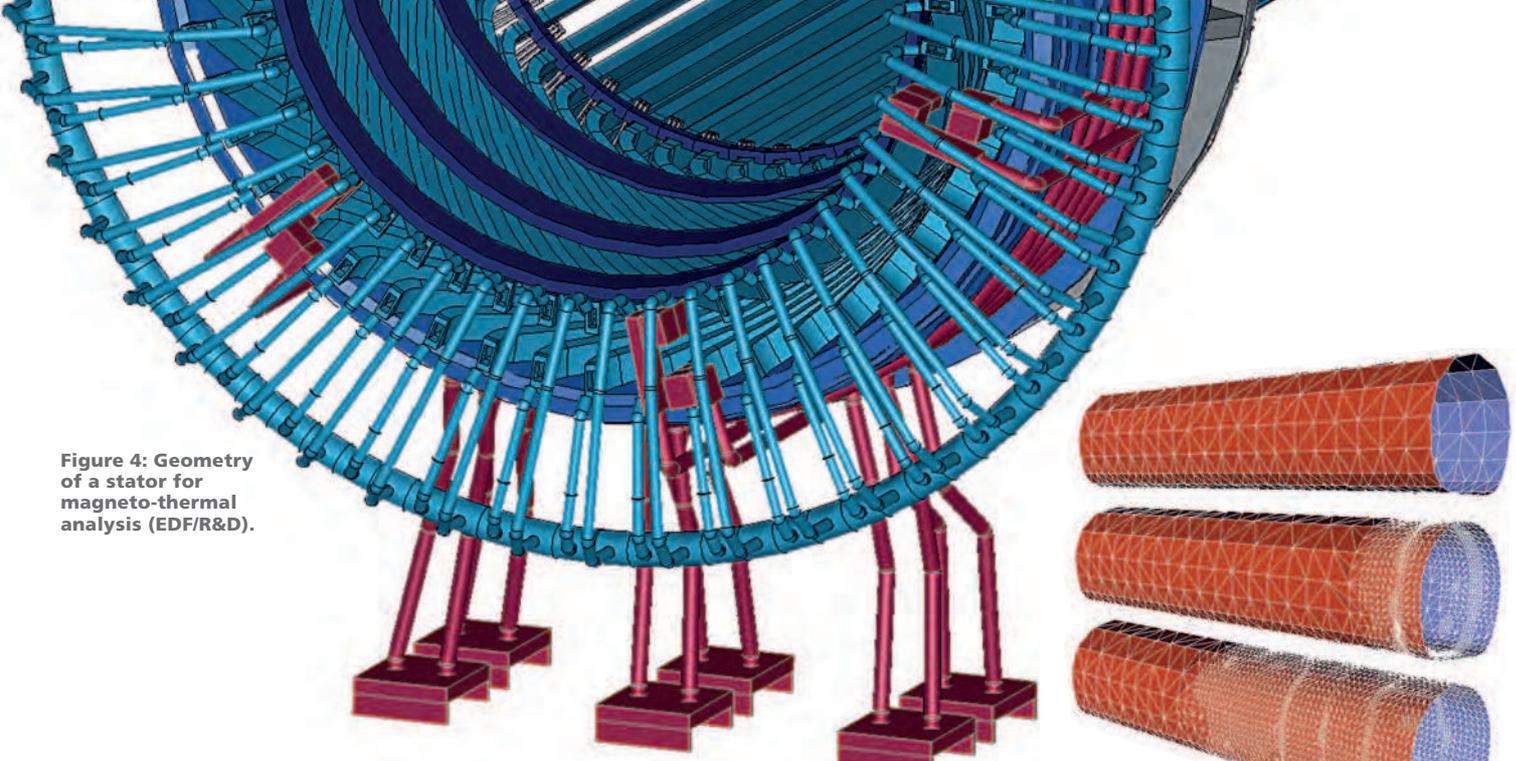


Figure 4: Geometry of a stator for magneto-thermal analysis (EDF/R&D).

### Optimization and refinement

The Meshing module comes with a companion named the **HOMARD module** to perform specific mesh adaptation to better fit the simulation objectives, in term of accuracy and performance.

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

HOMARD® is designed to operate in association with 2D/3D elements 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 local 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.

All HOMARD® instructions can be executed either using the graphical user interface or through the python interface (Figure 8).

### Creating hexahedric models

The SALOME platform provides you with specific mesh algorithms that help you with the creation of complex geometric models with hexahedric mesh representations. These meshes are specifically required for some numeric solvers, for example studies that involves fluid dynamic solvers.

In this objectives, you can use the HEXOTIC plugin of the SMESH module. This plugin is a wrapper for the commercial mesh program MeshGems-Hexa (formerly known as HEXOTIC) edited by the Distene Company. This function can create automatically a volumic hexahedric mesh of a complex geometry without any cubic partition of the solids (Figure 9).

An alternative method is to use the **HEXABLOCK module** that helps you to define a topological description isomorphic to the real geometry and from which a volumic hexahedric mesh can be automatically generated (Figure 10).

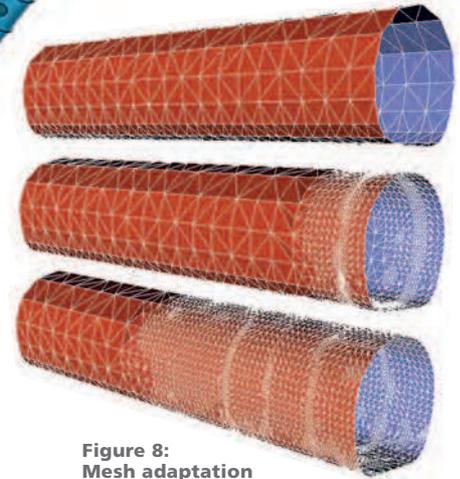


Figure 8: Mesh adaptation during the simulation of a tunnel excavation (EDF/R&D).

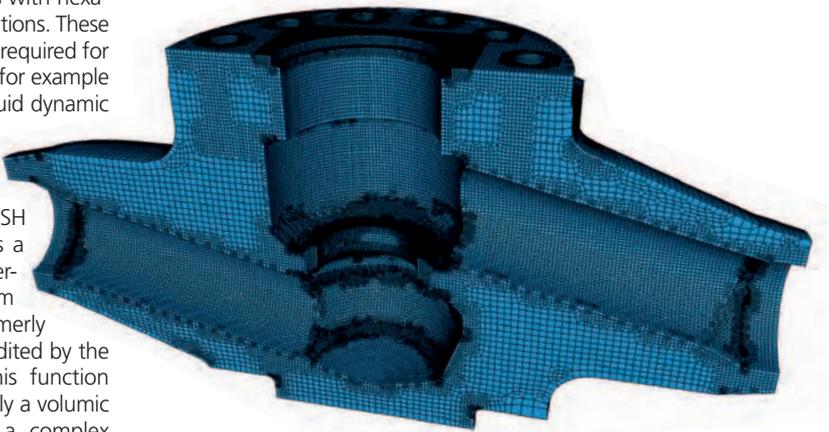


Figure 9: Mesh representation of a piping valve with automatic generation of hexahedron cells (EDF/R&D).

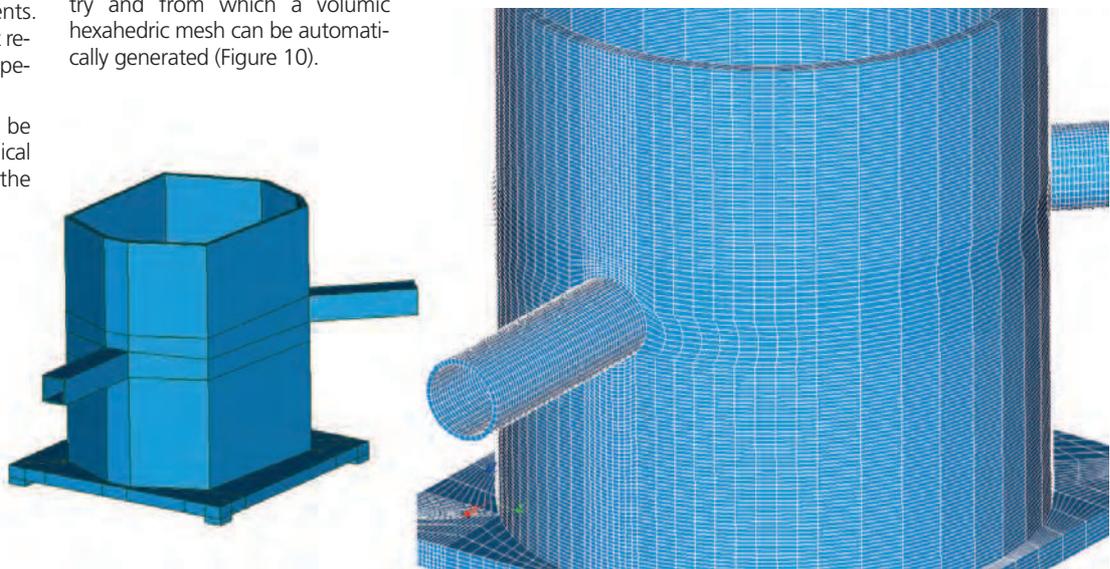


Figure 10: The topologic model of blocks (left), and the computed associated mesh (right) (CEA/DEN).

# SUPERVISION OF COMPUTATION WORKFLOWS

There is an increasing need for multidisciplinary parametric simulations in various research and engineering domains. Fluid-structure interaction and thermal coupling are two examples. The software strategy in many contexts of simulation (at least at CEA and EDF) is to develop numerical solvers dedicated to their own domain, and then to execute multi-domains simulation by coupling the existing dedicated solvers (Figure 11).

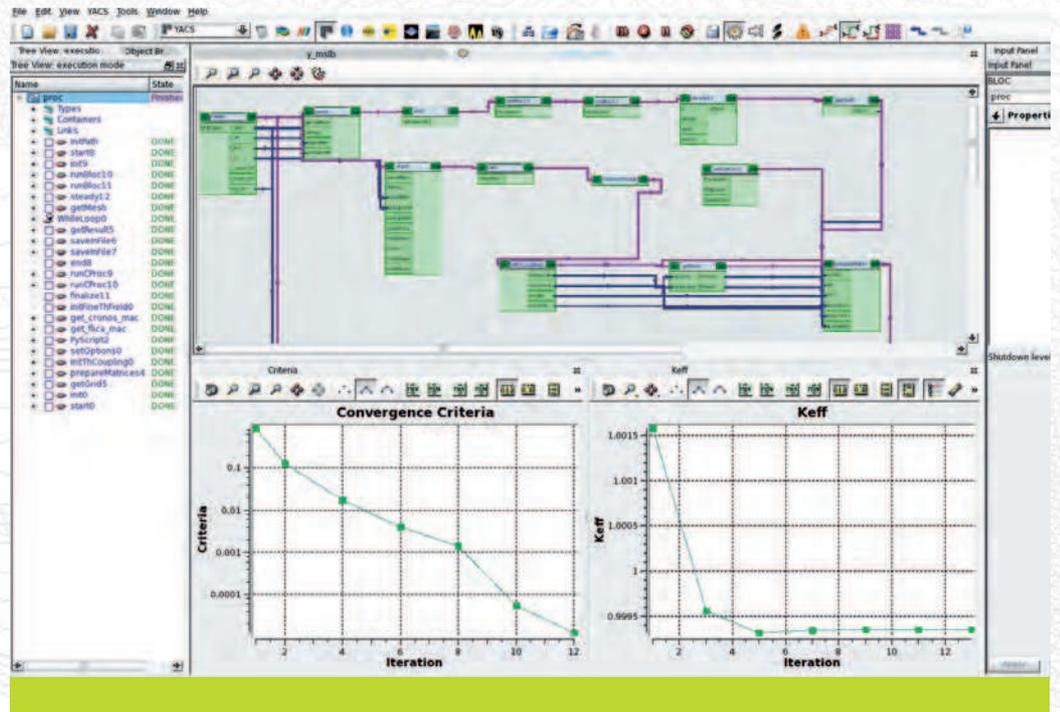


Figure 11: Coupling of a neutronics model with a thermalhydraulics model for a nuclear safety study (CEA/DEN).

The SALOME platform provides you with a set of services to create a simulation workflow that connects different computation units and then to execute the workflow on a distributed network of computers and HPC resources:

- The **YACSGEN program** helps you with the generation of a SALOME component that integrates your specific solver or any processing function.

The SALOME components are the computation units of a workflow designed with YACS.

- The **YACS module** contains the core set of services to compose the computation workflow as a graph of connected SALOME components.

This graph can be edited using a graphical user interface (GUI) or the python Text User Interface (TUI) to handle complex workflow into scripts.

- The **JOBMANAGER module** can be used to define a computation job (including either a simple SALOME component or a complete YACS workflow) and to drive the submission of the job to a distributed set of computers or HPC resources.

The module can handle the batch systems PBS, LSF, SGE, LOADLEVELER or SLURM through a normalized generic interface. It comes with a graphical user interface, but can be used at a programming level using a python interface.

Based on these core services dedicated to the workflows edition and supervision, you will find high level services to help you in the design of a numeric experimental plan:

- The **PARAMETRIC module** can be used to generate a parametric simulation, by exploration of a specified domain of the simulation parameters space.

# ANALYSIS OF SIMULATION DATA

## PROCESSING FIELDS AND MESHES

In SALOME, the MED data model defines a normalized representation to describe meshes of the geometry and fields of the physical values of the simulation. This data model is a key feature of the platform. It comes with a software implementation (the MED-file library) for file persistence and intercommunication of simulation data between components. SALOME meshing and visualization components propose interfaces with MED format to import or export data. Domain specific solvers that interact with SALOME are advised to use MED format for input and output files. This configuration leads to a high level of associativity between the software components to build simulation workflows.

Based on the concepts from the MED data model, the **MED-Coupling module** of SALOME provides you with a full set of services for high performance data processing on meshes and fields. Extensively rewritten for SALOME 7, the module contains the data structure to describe meshes and fields as C++ or python objects (MED-Coupling package). It provides a set of functions to manage the persistence toward the med file format (MEDLoader package), and functions to process the field data in standard algebraic operations (post-processing) or through interpolation and localization algorithms (INTERP\_KERNEL and REMAPPER packages), for example to perform field projections from a mesh to another (Figure12).

## VISUALIZATION OF SIMULATION RESULTS

The solvers interfacing with SALOME generate results that can be analysed within the ParaViS module. This module has been developed by integrating ParaView (third party software edited by Kitware on basis of the VTK toolkit) 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 analysed 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 easily obtained.

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 analyse large datasets.

(Figure 13).

## PROCESSING SIMULATION DATASET

Beyond the processing and visualization of meshes and fields data, the SALOME platform contains additional modules dedicated to advanced data analysis :

- **Data assimilation and optimisation environment**, for example to recalibrate the parameters of a model by comparison of the simulation data to experimental measures (module ADAO)
- **Propagation of uncertainties** in the simulation workflow, for example to evaluate the uncertainty on resulting data of the simulation considering a given uncertainty on the input parameters (module OPENTURNS).
- **Design numeric experimental plans**, for example a parametric simulation by exploration of a specified domain of the simulation parameters space (module PARAMETRIC).

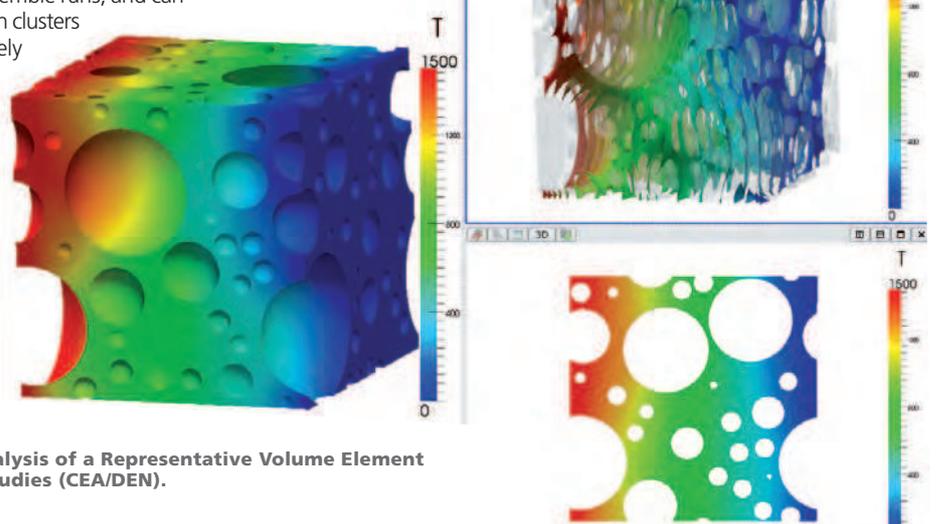


Figure 13: Analysis of a Representative Volume Element in material studies (CEA/DEN).

Figure 12: Projection of a field from a regular grid towards an unstructured mesh.

## CONTACT US:

Francis Kloss      francis.kloss@cea.fr  
Guillaume Boulant      guillaume.boulant@edf.fr  
Mikhail Kazakov      mikhail.kazakov@opencascade.com



Commissariat à l'Énergie Atomique  
et aux Énergies Alternatives,  
CEA-Saclay, DEN, DM2S,  
91191 Gif-sur-Yvette Cedex, France  
[www.cea.fr](http://www.cea.fr)



EDF R&D  
1, avenue du Général de Gaulle,  
92141 Clamart Cedex, France  
[www.edf.com](http://www.edf.com)



Open Cascade  
Filiiale d'EURIWARE, groupe AREVA,  
1 place des frères Montgolfier  
78044 Guyancourt, France  
[www.opencascade.com](http://www.opencascade.com)