



# SALOME9

THE OPEN SOURCE  
INTEGRATION  
PLATFORM  
FOR NUMERICAL  
SIMULATION



# SALOME9 PLATFORM

The SALOME platform is an open source software framework for the integration of numerical solvers in various scientific domains. CEA and EDF are using SALOME to perform a wide range of simulations, which are typically related to industrial equipment in power plants (nuclear power plants, wind turbines, dams...). Some primary concerns are the design of new-generation reactor types, nuclear fuel management and transport, material ageing for the life-cycle management of equipment, and the reliability and safety of the nuclear facilities.

To address these challenges, SALOME is integrating a CAD/CAE modeling tool, industrial mesh generators, and advanced 3D visualization features.



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



## KEY FEATURES

In order to accurately simulate complex industrial systems, scientists and engineers need to integrate domain specific solvers regarding material science, solid mechanics, structural dynamics, fluid physics, thermohydraulics, nuclear physics, radiations or electromagnetism. The SALOME platform provides one single simulation environment for all these fields.

The main features of SALOME are:

- **Design of the geometric representation** for physical systems (CAD modeling) and its associated discretized model (meshing functions for finite elements or finite volumes solvers).
- **Ability to integrate domain specific solvers** into normalized software components with standard interfaces to facilitate the coupling of different physical domains.
- **Supervision of computation workflows** defined as graphs of distributed software components, including CAD modeling, domain specific solvers and data processing components.
- **Analysis of simulation output** in particular using visualizations of physical fields resulting from computation workflows in 3D views or in plot charts.

In this context, one of the key points of the platform is the usage of standardized data models to describe physical concepts for numerical analysis, and to ensure interoperability between software components. For instance, the MED data model is used for meshes and fields descriptions.

## MODEL OF DEVELOPMENT

The SALOME platform is actively developed by CEA and EDF, two key players in the French energy industry, and with the support of the Capgemini subsidiary OPEN CASCADE, one of the leaders in software development for scientific computing. This 15 years old partnership provides the SALOME project with a very committed and dedicated team specialized in scientific computing. Moreover, the SALOME platform strongly relies on the integration of cutting edge third party software programs: the commercial advanced and robust meshing programs MeshGems-CAD-Surf and MeshGems-Tetra (by DISTENE S.A.S.) and ParaView including the VTK 3D visualisation toolkit (by Kitware Inc.).

## DOWNLOAD THE SALOME PLATFORM

The SALOME platform is available under LGPL licence and can be downloaded from the web site <http://www.salome-plateform.org> for several LINUX distributions and Windows. The site provides tutorials, a forum section and gives access to the user documentation and the source code.

## SERVICE AND SUPPORT

OPEN CASCADE provides a whole range of services around SALOME for professional end-users, such as technical support and specific training.

The "à-la-carte" support program is particularly suited for universities and academic organizations, as well as industrial companies:

- Helpdesk and technical support for expert needs concerning a one-shot technical issue, delivered by mail or by phone within a guaranteed time frame.
- Expert consulting delivered on the end-user premises by one of the SALOME experts.
- Bug corrections and improvements for specific needs or complex problem solving.
- Creation of geometries and meshes (geometric modeling, meshing, data input) for the execution of numerical simulation studies.
- SALOME training allows most of fundamental and advanced functionalities of the platform to be mastered.

**For more details, consult:**  
<http://www.salome-plateform.org/service-and-support>

## DEVELOPMENT FACT SHEET

**Project kick-off:** 2001

**Development team:** 20 engineers

**Licence:** LGPL

**Distribution:** Most Linux distributions and Windows operating systems

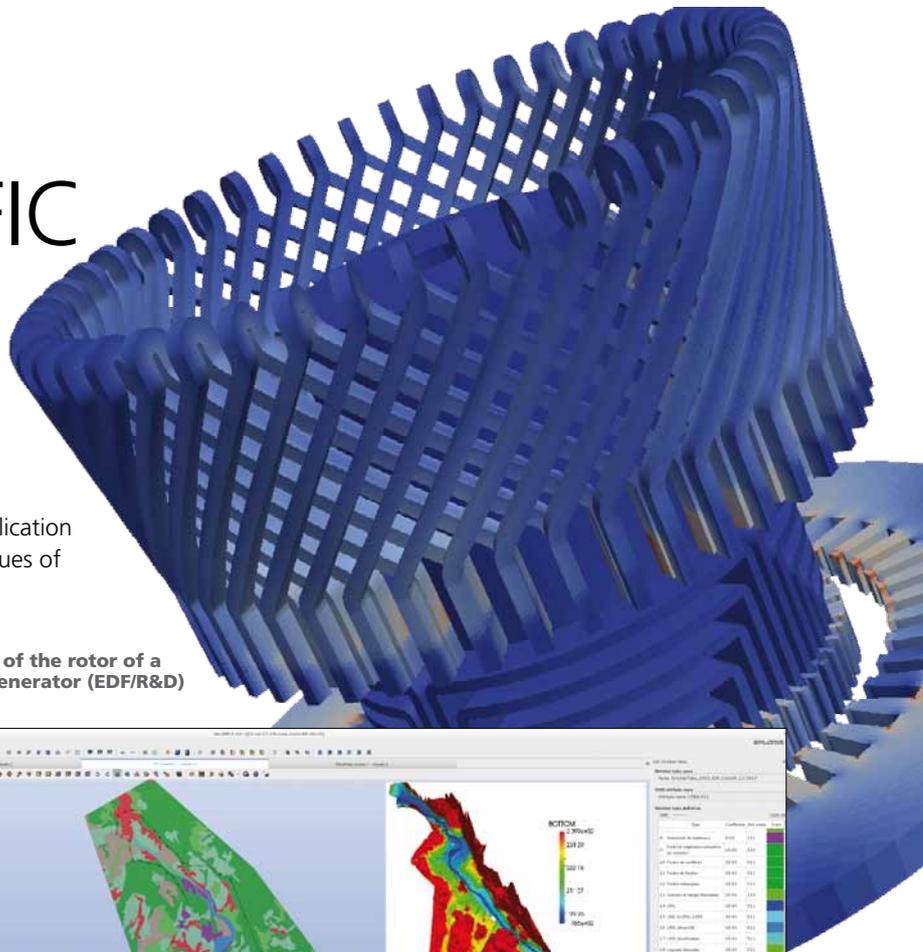
**Bug tracking:** 500 fixes and improvements per year

**Verification & Validation:** automated test procedure with 6000 tests

**Users:** 400 users at EDF and CEA, 144000 downloads in 2018

# DEVELOPING DOMAIN SPECIFIC APPLICATIONS

The SALOME platform allows the integration of specific solvers and graphical user interfaces to create engineering-specific simulation applications. For example, Salome\_Meca provides a simulation environment for solid and structure mechanics, based on the code\_aster solver. At EDF, this application is used to study ageing equipments, reliability and safety issues of nuclear vessels, turbine vibrations and much more.



Analysis of the rotor of a power generator (EDF/R&D)

The SALOME-HYDRO platform is an example of application dedicated to free-surface hydraulic analysis. The screenshot (see Figure 2) illustrates modeling of a river from bathymetric data using the platform:

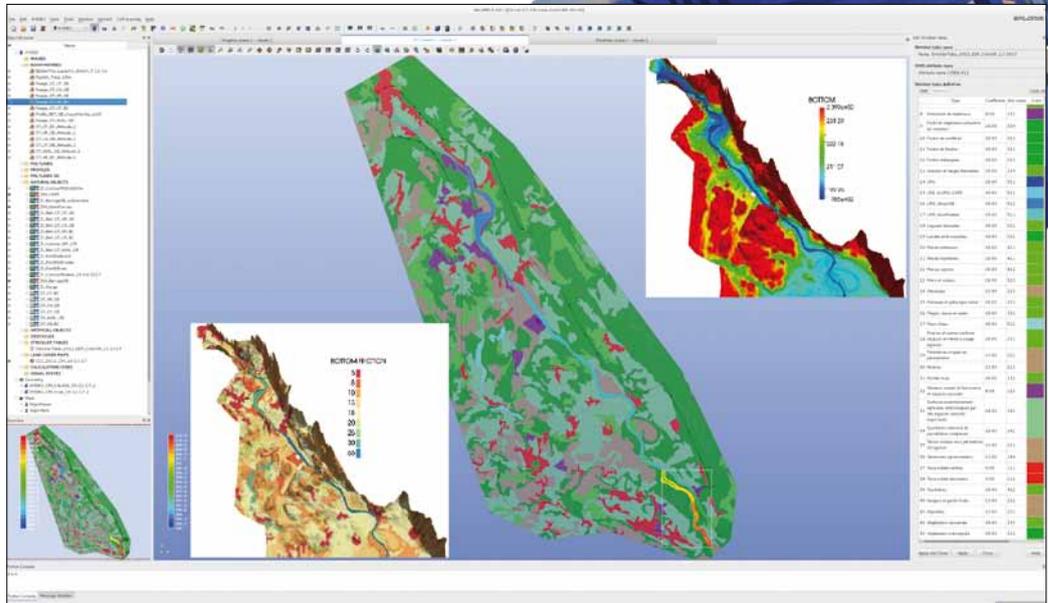


Figure 2: River model from bathymetric data using SALOME-HYDRO (EDF/R&D)

Another example (see Figure 3) is a SALOME based application named ALAMOS, which is dedicated to the modeling of nuclear reactor for thermo-nuclear analysis.

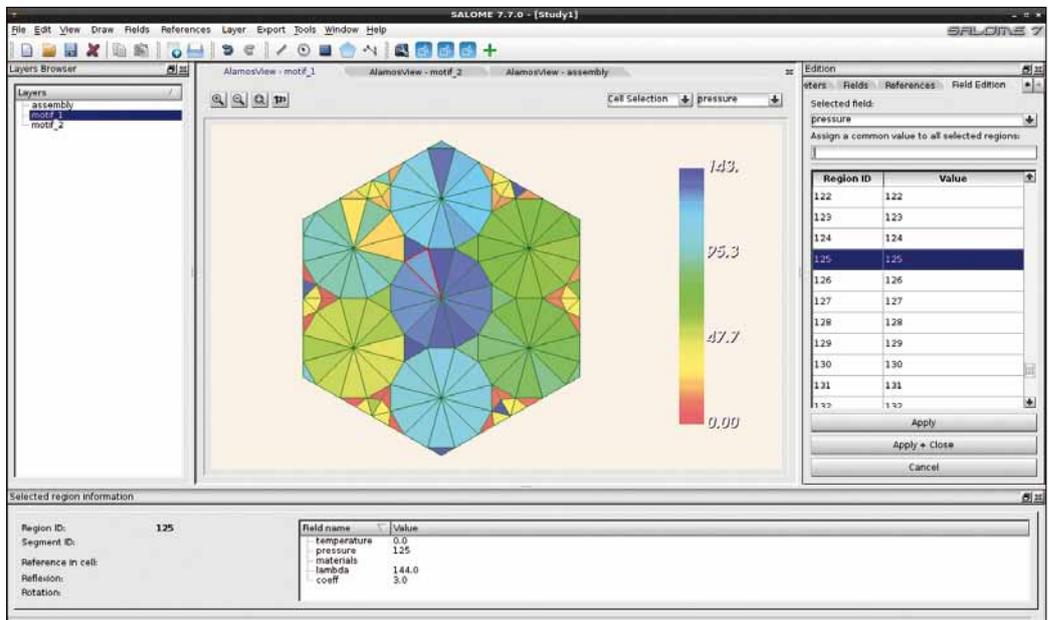


Figure 3: Illustration of the ALAMOS application dedicated to the modeling of nuclear reactors (CEA/DEN)

In this context, SALOME is distributed with the same quality assurance as any industrial process, including Verification & Validation. The development team provides top-level services and support to build dedicated applications.

# MODELLING PHYSICAL SYSTEMS



SALOME provides several modules to create complex geometrical models. These include high level meshing functionalities to prepare numerical models that fit the solver requirements.

## GEOMETRIC MODELING

Shaper is the new CAD modeler of SALOME (available in SALOME 9.3). SHAPER is a parametric and variational CAD modeler. It offers improved ergonomics that are best suited for the drawing from the graphical interface. The variational aspect allows the user to draw interactively a sketch with dimensions and constraints, just what he sees on a technical drawing. The parametric aspect ensures that the final shapes are automatically updated when a parameter of a CAD feature is modified. It also allows simple assembly of 3D parts.

SHAPER has been developed specifically for simulation CAD models:

- shape design is dedicated to the creation of conformal meshes;
- definition of groups of shapes is available and automatic update of groups is done if the geometry is modified;
- connection of multi-dimensional geometries can be done;
- non manifold geometries are feasible, like for example an edge shared by more than two faces;
- geometries may be designed using the Text User Interface (TUI) through Python scripts.

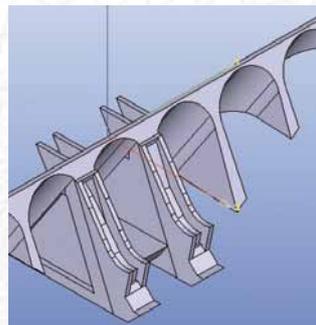
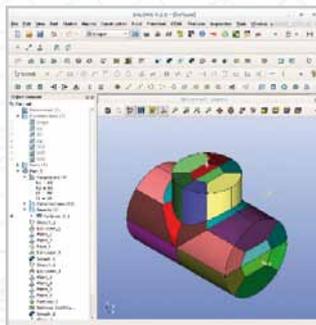
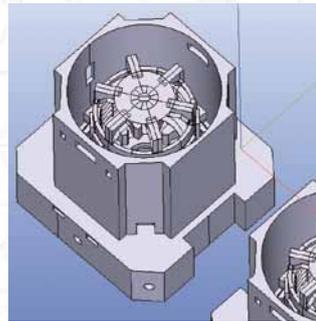
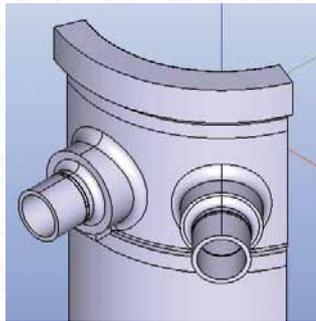


Figure 4: Examples of industrial geometries (EDF/R&D)

## MESHING THE GEOMETRY

The meshing module known as SMESH provides a wide range of algorithms particularly suited for finite-element and finite-volume methods. A mesh can be divided into groups to segregate different regions of the geometry. It allows differentiation between mesh properties or even between mesh types (hexahedral or tetrahedral). Group naming provides the identification of local boundaries and initial conditions, and facilitates the mesh visualization or other post-processing operations (see Figure 4).

### Mesh module

A complete toolbox enables the user to check the mesh quality and to perform local modification or adjustment. Transformation operations can be used to produce complex meshes or compounds.

The meshing process can be entirely handled by Python scripting to ensure full reproducibility and parameterization of the simulation workflow.

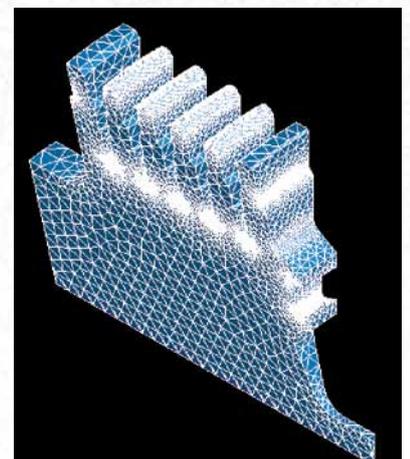
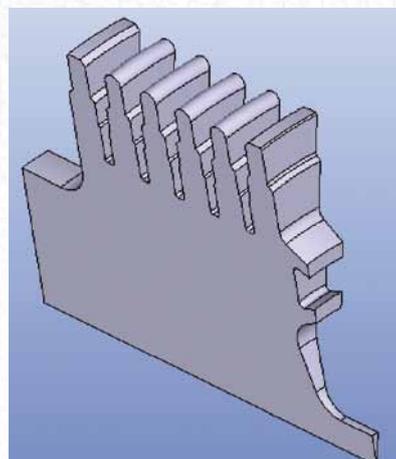
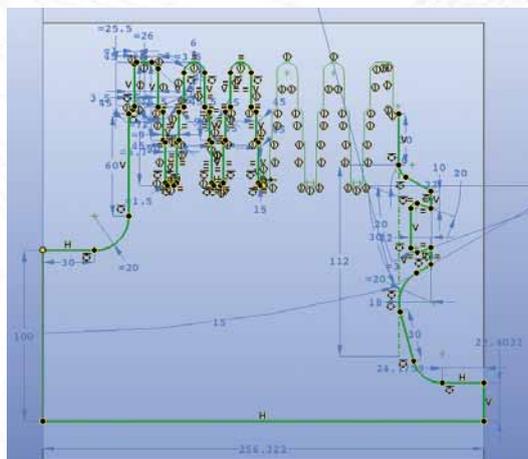


Figure 5: Example of industrial geometry from the sketch to the mesh



### Catalog of meshing algorithms

The SALOME platform integrates both open-source meshing tools such as NETGEN (<https://sourceforge.net/projects/netgen-mesher>) and GMSH (<http://gmsh.info>), and commercial ones such as MeshGems-CADSurf and MeshGems-Tetra edited by DISTENE S.A.S ([www.distene.fr](http://www.distene.fr)).

These powerful meshing tools are based on different algorithms and properties (local size, growth rate, enforced vertices...) that can be adjusted to get the best mesh quality for each specific numerical simulation.

These different meshing algorithms can be combined in SALOME. For example, the 2D trans-patch meshing algorithm from GMSH can be used for the boundary surface mesh together with the 3D MeshGems-Hybrid algorithm for the internal volume (see Figure 6).

### Hexahedral meshes

SALOME provides specific mesh algorithms that help to create complex models with hexahedral mesh representations. This kind of mesh is specifically required for some numerical solvers, for example studies involving fluid dynamics solvers.

SALOME provides several tools to solve this problem. The HEXABLOCK module can be used to define a topological description isomorphic to the real geometry and from which a hexahedral mesh can be automatically generated. The MeshGems-Hexa SMESH plugin is a wrapper of the commercial mesh program edited by the DISTENE S.A.S. This function can create automatically a hexahedral mesh of a complex geometry without any partition of the solids.

### Hybrid meshes

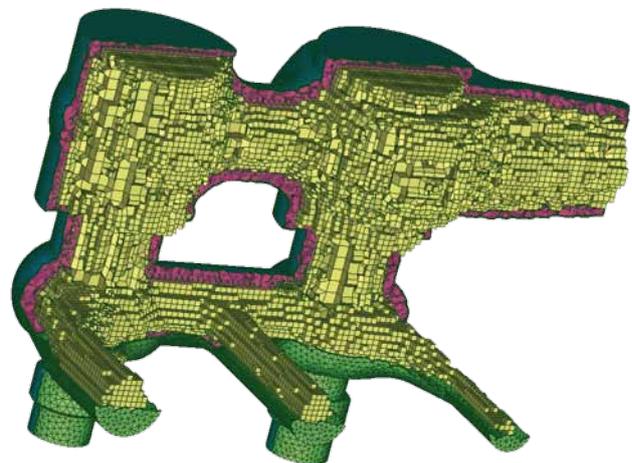
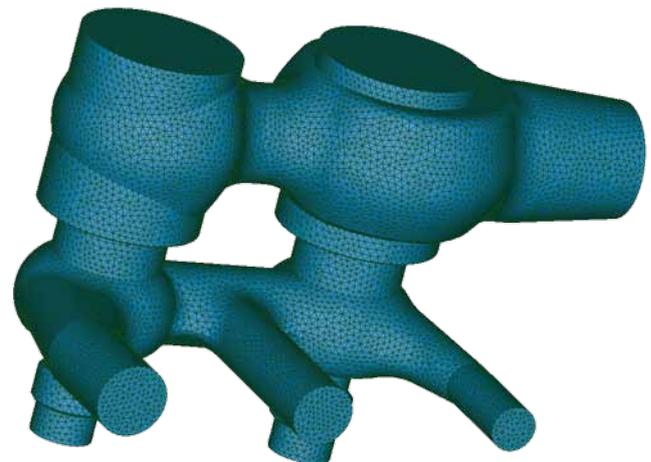
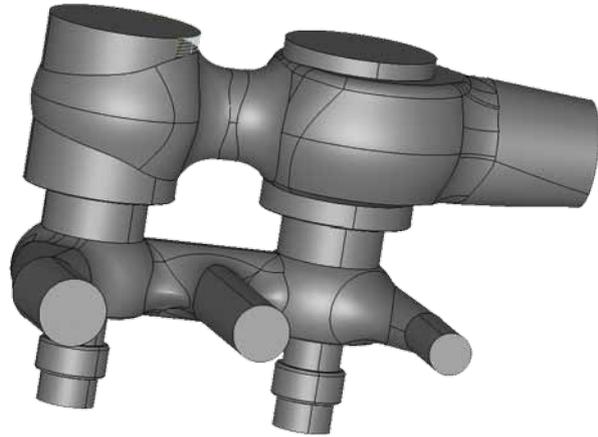
SALOME provides a specific mesh algorithm to create hybrid meshes (see Figure 6), mixing tetrahedra and hexaedra. This tool is based on the MeshGems-Hybrid component (from DISTENE S.A.S.).

MeshGems-Hybrid generalizes the automatic conformal volume filling for all closed tri-quad surfaces given as an input, by mixing together the methods of MeshGems-Tetra and MeshGems-Hexa in addition to an extrusion method. In addition, as a subset of the whole set of MeshGems-Hybrid capabilities, boundary layers can be generated automatically ([www.meshgems.com/volume-meshing-meshgems-hybrid.html](http://www.meshgems.com/volume-meshing-meshgems-hybrid.html)).

### Optimization and refinement

SMESH is completed with the HOMARD<sup>®</sup> module that performs local mesh adaptations required by numerical codes to meet accuracy and performance.

To improve the quality of the simulations, local mesh adaptations offer an efficient compromise between a fine mesh and a low computational cost. HOMARD<sup>®</sup> allows refinement and coarsening operations to adapt the mesh, according to the numerical error of the simulation.



**Figure 6: SALOME can combine different meshing algorithms. In this example, the 2D trans-patch meshing algorithm from GMSH is used for the boundary surface mesh, while the 3D MeshGems-Hybrid algorithm is used for the internal volume to get a globally hexahedral mesh**

# 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-domain simulations by coupling these specific solvers.

SALOME provides a set of services to create a simulation workflow that connects different computation units. Then it executes this workflow on a distributed network of computers and HPC resources.

The main features are:

- The possibility to integrate domain specific solvers** as normalized components with standard interfaces to ease the coupling of different physical domains. These SALOME components can be used as the computational units of a simulation process. Some tools are provided to automatize this integration for standard configurations (integration of executable programs, functions of a library or python scripts).

- The supervision of a computation workflow** defined as a graph of connected SALOME components, including CAD modeling, meshing, domain specific solvers and data processing components (see Figure 7). The graph can be edited using a graphical user interface (GUI) or the Python Text User Interface (TUI) to handle complex workflow as scripts.

- The design of numeric experiments.** SALOME provides a scheduler that helps the user to manage the parametric computation (see Figure 8). The input data are the experimental plan (typically csv data) and the computation unit (deterministic function). It can be used together with advanced modules such as OpenTURNS to generate the input sample and analyze the output results (meta-modeling, statistical analysis, uncertainty quantification).

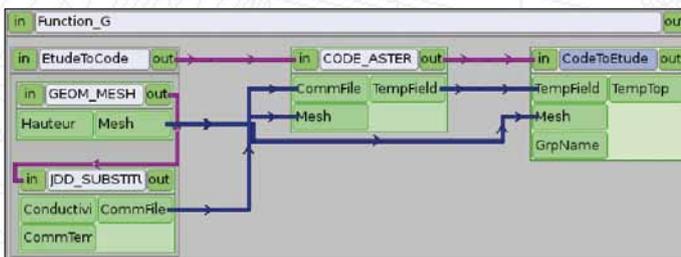


Figure 7: Typical example of a simple computing data flow including CAD, mesh, physical solver and data postprocessing (EDF/R&D)

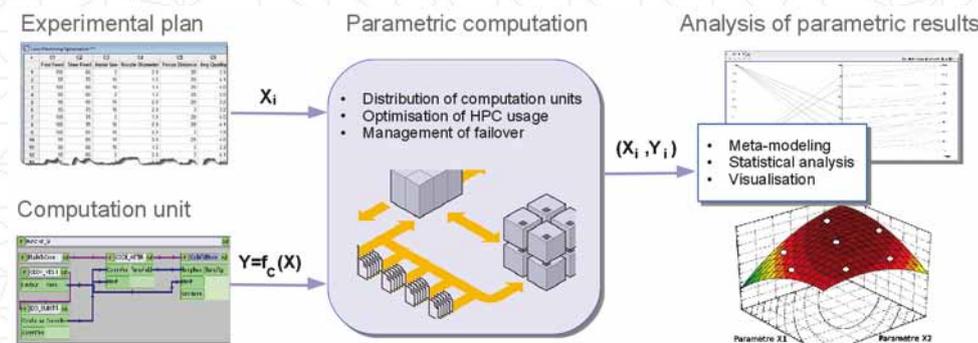


Figure 8: Design of numeric experimental plans and distribution of computation units on HPC with failover management (EDF/R&D)

# ANALYSIS OF SIMULATION DATA

## PROCESSING FIELDS AND MESHES

Inside SALOME, the MED data model defines a normalized representation to describe meshes on the geometry and fields of the physical values of the simulation. This data model is a key feature that helps specific solvers to take advantage of SALOME services.

It comes with a software implementation (the MED-file library) for file persistence and serialization in memory for inter-communication of simulation data between components.

Based on concepts from the MED data model, the **MED module** provides a full set of services for high performance data processing. It includes the MEDCoupling package which implements meshes and fields C++ classes fully wrapped in Python. MEDCoupling also provides a large set of methods to handle efficiently in memory all fields and meshes objects (interpolation, algebraic operations, field extractions, integration...) and allows to easily exchange fields and meshes between processes (directly in memory or using files).

## VISUALIZATION OF SIMULATION RESULTS

Physical solvers generate results that can be analysed within the ParaViS module. This module has been developed to integrate ParaView (third party open source software edited by Kitware and based on the VTK toolkit) into SALOME, and to offer all the functionalities of this award-winning post-processor tool.

A wide range of representations are available to explore the datasets: surface, volume, gauss points... The data can then be analysed using many filters to extract significant data: clip, threshold, iso-surface, stream lines, elevation surfaces (see Figure 9).

Quantitative information can be extracted using the analysis tools: selection of the data, histograms, plots over time or curvilinear abscissa are one click away.

All these features can be animated within the module to analyse time-varying data, sweep a cutting plane through the dataset, or animate the deflection shapes of a modal analysis (see Figure 10).

This module is fully scriptable in Python to create visualizations in batch when necessary or to repeat analysis on parametric runs. It is used on remote visualization clusters to interactively analyse large datasets (see Figure 11).

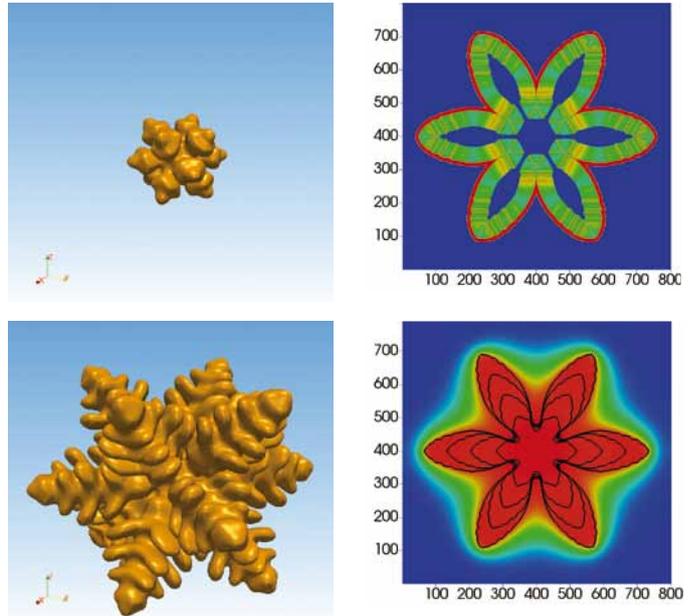


Figure 10: Crystal growth simulation using the Lattice Boltzmann equation (CEA/DEN)

## PROCESSING SIMULATION DATASETS

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

- **Data assimilation and optimization environment**, for example to recalibrate the parameters of a model by comparison of the simulation data to experimental measures (modules ADAO, URANIE).
- **Propagation of uncertainties** in the simulation workflow, for example to evaluate the uncertainty of the resulting data considering a given uncertainty on the input parameters (modules OpenTURNS, URANIE).

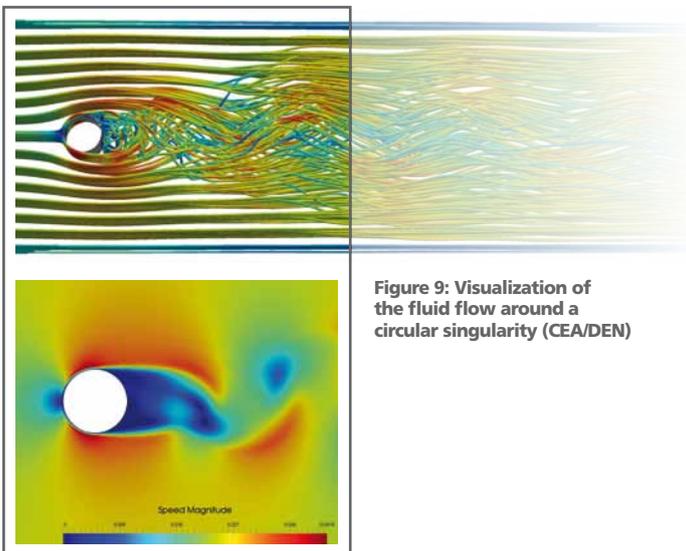


Figure 9: Visualization of the fluid flow around a circular singularity (CEA/DEN)

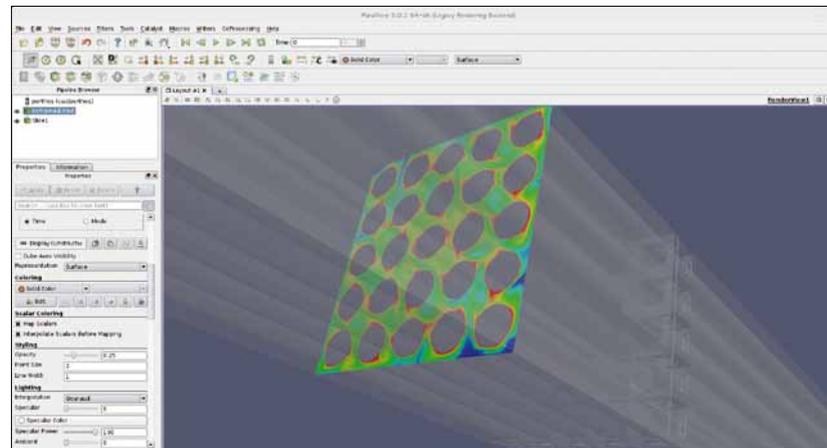


Figure 11: Visualization of thermohydraulics results using PARAVIS in remote mode (EDF/R&D)

2019



[salome-platform.org](http://salome-platform.org)

## CONTACT US:

Francis Kloss      francis.kloss@cea.fr  
Vincent Stobiac      vincent.stobiac@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  
EDF Lab Paris-Saclay  
7, boulevard Gaspard Monge  
91120 PALAISEAU, France  
[www.edf.com](http://www.edf.com)



Open Cascade  
groupe Capgemini,  
1 place des frères Montgolfier  
78044 Guyancourt Cedex, France  
[www.opencascade.com](http://www.opencascade.com)