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...). Among 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 modelling tool, industrial mesh generators, and advanced 3D visualization features.

KEY FEATURES
In order to accurately simulate complex industrial systems, scientists and engineers need to integrate most fields of physics such as material science, solid mechanics, structural dynamics, fluid physics, thermodynamics, nuclear physics, radiations or electromagnetism. The SALOME platform gathers all these fields in one single simulation environment.

The main features of the SALOME are:
- Design of the geometric representation for physical systems (CAD modelling) 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 modelling, 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 SALOME project with a very committed and dedicated team specialized in scientific computing.

Moreover, the SALOME platform heavily relies on the integration of cutting edge third party software programs: the commercial advanced and robust meshing programs MeshGems-CADSurf and MeshGems-Tetra (by DISTENE S.A.S.) and ParaView including the VTK 3D visualisation toolkit (by the 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-platform.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 & 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 & improvements for specific needs or complex problem solving.
- Creation of Geometries and Meshes (geometric modelling, Meshing, data input) for the execution of numerical simulation studies.

SAIOME Training allows most of fundamental and advanced functionalities of the platform to be mastered.

For more details, consult: http://www.salome-platform.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 6,000 tests
Users: 300 users at EDF and CEA, 50,000 downloads this year
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 solvers. At EDF, this application is used to study ageing equipments, reliability and safety issues of nuclear vessels, turbine vibrations and much more.

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.

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

In this context, SALOME is distributed with the same Quality Assurance as any industrial process, including Verification and 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 MODELLING

The CAD module (known as GEOM) provides a rich set of functionalities to create, edit, import or modify CAD models. The geometric shapes may be designed interactively using the Graphical User Interface (GUI) or the Text User Interface (TUI) through Python scripts. All GEOM functionalities are available in both GUI and TUI. This allows complex shapes or several configurations of a shape with different values of its parameters to be built. This enables the user to parameterize its geometry and play different scenarios without any effort.

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

Figure 4: Modelling of the rotor of a power generator (EDF/R&D)

Figure 5: Modelling of the volume around a circular singularity for fluid flow analysis (CEA/DEN)

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. Like the CAD process, the meshing process can be entirely handled by Python scripting to ensure full reproducibility and parameterization of the simulation workflow.
Catalog of meshing algorithms

The SALOME platform integrates either open-source meshing tools such as NETGEN (https://sourceforge.net/projects/netgen-mesher) and GMSH (http://gmsh.info), or commercial ones such as MeshGems-CADSurf and MeshGems-Tetra edited by DISTENE S.A.S (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 illustration below).

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 HEXA-BLOCK 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 hexahedra. This tool is based on the MeshGems-Hybrid component (from DISTENE S.A.S.).

MeshGems-Hybrid is generalizing 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. And, 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).

Optimization and refinement

SMESH is completed with the HOMARD® 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® 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-domains simulation 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 modelling, 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 into scripts.

- The distribution on HPC resources. SALOME contains a job manager that can be used to define a computation job (including either a simple SALOME component or a complete workflow) and to drive the submission of the job to a distributed set of computers or HPC resources. The job manager can handle many batch systems like PBS, LSF, SGE, LOADLEVELER or SLURM through a normalized generic interface. It comes with a GUI but can be used at a programming level using a C++ or Python interface to create simple scripts or domain specific tools.

- The design of numeric experimental plans. SALOME provides you with 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 that helps you design the input experimental plan and the analysis of output results (meta-modelling, 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 of 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 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 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 data analysis tools: taking a 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).

PROCESSING SIMULATION DATASET
Beyond the processing and visualization of meshes and data fields, 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 (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 10: Crystal growth simulation using Lattice Boltzmann equation (CEA/DEN)

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