

SALOME 6.6.0

Minor release announcement

December 2012



GENERAL INFORMATION

CEA/DEN, EDF R&D and OPEN CASCADE are pleased to announce [SALOME](#) version [6.6.0](#). It is a minor release that contains the results of planned major and minor improvements and bug fixes against SALOME version 6.5.0 released in June 2012.

Table of Contents

- **GENERAL INFORMATION1**
- **NEW FEATURES AND IMPROVEMENTS4**
 - PREREQUISITES CHANGES 4
 - LICENSE RESTRICTIONS 5
 - MAIN IMPROVEMENTS 6
 - Viscous Layers 2D* 6
 - Reorient faces by vector* 6
 - Fuse Collinear Edges* 7
 - Get non blocks* 7
 - Divided Disk and Divided Cylinder* 8
 - Restore Path* 9
 - Boolean Operations on Groups* 9
 - 0D Elements on Element Nodes* 10
 - Redesigned Mesh Properties dialog box* 10
 - HEXABLOCK MODULE 11
 - MED MODULE CHANGE LOG 11
 - API Modification in Python only* 11
 - API Modification in C++/Python* 11
 - Behavior modification* 11
 - New features* 12
 - Python interface enhancement* 13
 - Bugs solved* 13
 - DOCUMENTATION IMPROVEMENTS 13
 - OTHER IMPROVEMENTS 14
- **BUG CORRECTIONS16**
 - GUI MODULE (IAPP) 16
 - KERNEL MODULE 17
 - GEOM MODULE 18
 - SMESH MODULE 23
 - MED MODULE 30
 - YACS MODULE 30
 - PARAVIS MODULE 31
 - BLSURFPLUGIN MODULE 32
 - GHS3PLUGIN MODULE 33
 - HEXOTICPLUGIN MODULE 33
 - NETGENPLUGIN MODULE 33
 - INSTALLATION PROCEDURE 34

- OCCT 6.5.4 BUG CORRECTIONS35
- SUPPORTED LINUX DISTRIBUTIONS AND PRE-REQUISITES38
- HOW TO INSTALL AND BUILD SALOME41
- SALOME SYSTEM REQUIREMENTS41
- HOW TO GET THE VERSION AND PRE-REQUISITES41
- KNOWN PROBLEMS AND LIMITATIONS42



NEW FEATURES AND IMPROVEMENTS

PREREQUISITES CHANGES

Important: SALOME 6.6.0 is based on the Open CASCADE Technology version 6.5.4. The source code of SALOME is no more compatible with Open CASCADE Technology 6.5.3 and older versions.

The table below provides the full list of pre-requisite products used with SALOME 6.6.0. The table shows the differences of 3rd-party product versions used for SALOME 6.6.0 and previous release 6.5.0; the changes are highlighted in bold.

Product	SALOME 6.5.0	SALOME 6.6.0
Boost	1.49.0	1.49.0
Cgns	3.1.3-4	3.1.3-4
CMake	2.8.7	2.8.7
Docutils	0.8.1	0.8.1
Doxygen	1.8.0	1.8.0 ¹
Expat	2.0.1	2.0.1
Freeimage	3.14.1	3.14.1
freetype	2.3.7	2.4.10
Ftgl	2.1.2	2.1.3-rc5
gl2ps	1.3.5	1.3.5
Graphviz	2.28.0	2.28.0
HDF5	1.8.8	1.8.8
Homard	10.4	10.5
Intel® Threading Building Blocks	3.0	3.0
Jinja2	2.6	2.6
LAPACK	3.3.0	3.3.0
libBatch	1.5.0	1.6.0
Libxml2	2.7.8	2.7.8 ²
Med	3.0.5	3.0.6
METIS	4.0	4.0
NETGEN	4.9.13	4.9.13
NumPy	1.5.1	1.5.1
omniORB	4.1.6	4.1.6
omniORBpy	3.6	3.6
omniNotify	2.1	2.1
Open CASCADE Technology	6.5.3	6.5.4

¹ Patched for SALOME (bugs in doc tree javascript)

² Patched for SALOME (bug for 64bits platforms)

Product	SALOME 6.5.0	SALOME 6.6.0
ParaView	3.14.0	3.14.0 ³
Pygments	1.5	1.5
PyQt	4.9.1	4.9.1 ⁴
Python	2.6.6	2.6.6
QScintilla	2.6.1	2.6.1 ⁵
Qt	4.6.3	4.6.3
Qwt	5.2.1	5.2.1
Scotch	5.1.11	5.1.11
Setuptools	0.6c11	0.6c11
SIP	4.13.2	4.13.2
Sphinx	1.1.3	1.1.3
SWIG	1.3.40	1.3.40 ⁶
Tcl	8.5.8	8.5.8
Tk	8.5.8	8.5.8
TclX	8.4.0	8.4.0
VTK ⁷	5.9.0	5.9.0
Xdata	0.9.3	0.9.6
Distene Blsurf ⁸	3.1	MeshGems suite v1.0.1
Distene TetMesh-GHS3D ⁹	4.2 + 4.1	MeshGems suite v1.0.1
Distene Hexotic ¹⁰	1.0	1.0

For additional information about pre-requisite products and SALOME modules dependencies refer to the paragraph "[Supported Linux distributions and pre-requisites](#)" below.

LICENSE RESTRICTIONS

- Hereby we explicitly declare that PyQt 4 toolkit (Riverbank Computing Ltd) is distributed under the terms of GPL license.

³ Patched (fix problems in build procedure with plug-ins and documentation)

⁴ Patched (fix issues of incompatibility with Qt 4.6.3)

⁵ Patched (fix issues of incompatibility with Qt 4.6.3)

⁶ SWIG 1.3.40 does not compile at latest versions of gcc (4.6); instead version 2.0.4 can be used

⁷ Included to the ParaView distribution

⁸ Commercial product, requires license for using in runtime

⁹ Commercial product, requires license for using in runtime

¹⁰ Commercial product, requires license for using in runtime

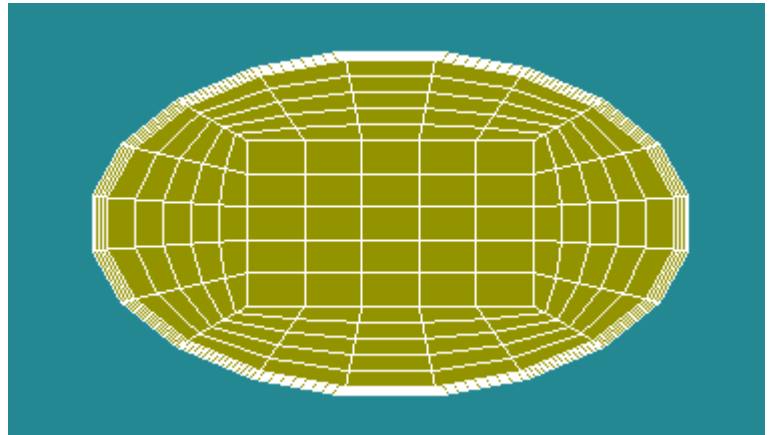
MAIN IMPROVEMENTS

Viscous Layers 2D

Viscous Layers 2D hypothesis is an additional hypothesis for creation of viscous layers on geometrical faces. It works with Triangle (MEFISTO), Quadrangle (Mapping), NETGEN 2D, NETGEN 1D-2D and BLSURF meshing algorithms.

The following parameters can be defined for the hypothesis:

- *Total thickness* - gives the total thickness of element layers;
- *Number of layers* - specifies the number of element layers;
- *Stretch factor* - specifies the growth factor of element height from the mesh boundary inwards.



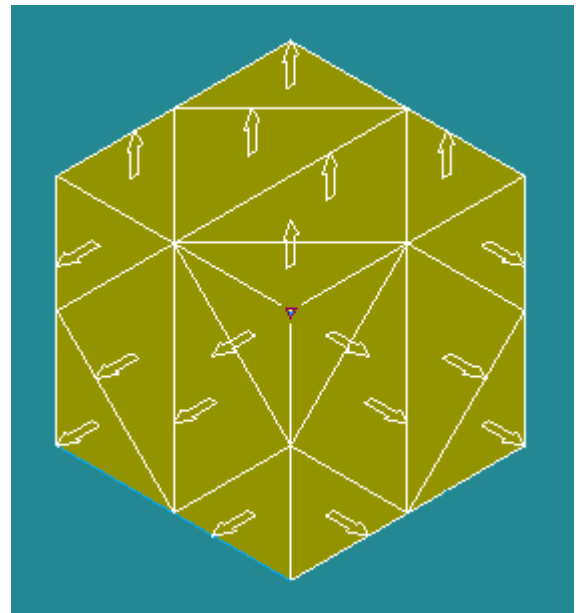
The image shows a circular face with viscous layers close to its border.

Reorient faces by vector

Reorient faces by vector operation allows changing the orientation of a set of neighboring faces. New orientation is defined by a vector.

To perform this operation in the main menu, select **Modification → Reorient Faces by Vector**.

In the dialog box it is possible to specify the Direction of the vector. It is also necessary to define the control face with the normal that will be compared with the vector. This face can be defined either explicitly or found by proximity to a given point.



Orientation of faces

Fuse Collinear Edges

Fuse Collinear Edges operation removes the selected vertices from a given wire, provided that the junction of two adjacent edges forming the wire is C1 continuous (i.e. the edges have equal tangents at the junction).

To perform this operation in the main menu, select **Repair** → **Fuse Collinear Edges within a Wire**.

The function takes a list of vertices to suppress as a parameter. If the list is empty, all vertices in a wire are taken into account.

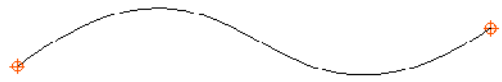
In Python this operation can be called using method

```
FuseCollinearEdgeswithinwire(thewire,
                               thevertices=[]).
```

Additionally the option **Fuse collinear edges to allow bigger radius** has been introduced in Fillet1D functionality to fuse collinear edges of the working wire automatically. It is available in Python via parameter `doIgnoreSecantVertices` of `MakeFillet1D()` function, by default it is *True*.



A wire formed by two collinear edges



The wire after Fuse Collinear Edges operation

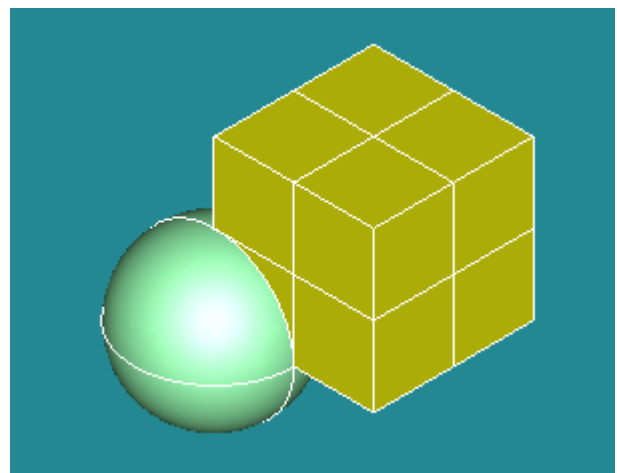
Get non blocks

New measurement **Get non blocks** returns a group of all non-block solids and a group of all non-quadrangular faces from the given shape (block solids are formed by six quadrangular faces).

The operation is available from the main menu via **Measures** → **Get non Blocks**.

Alternatively it is possible to use TUI function

```
(GroupNonBlocks, GroupNonQuads) =
    geompy.GetNonBlocks(shape)
```



Non-block sphere is cyan

Divided Disk and Divided Cylinder

Divided Disk is a disk divided into blocks and prepared for hexahedral meshing.

To create a divided disk in the main menu, select **New Entity** → **Advanced** → **Divided Disk**.

A divided disk can be defined by radius and orientation (plane "OXY", "OYZ" or "OZX") with origin at the center of coordinates or by coordinates of the disk center, normal to the disk plane and radius.

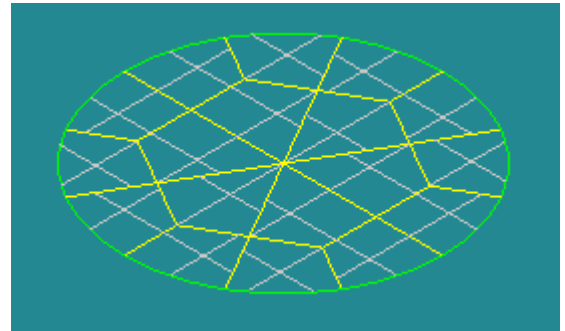
The disk is divided by a predefined square or hexagonal pattern (the latter provides a better mesh quality and especially less acute or obtuse angles)

In TUI this operation is available through commands

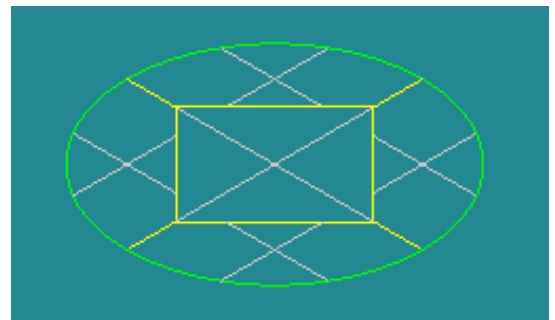
```
geompy.MakeDividedDisk(Radius, Orientation,
                        Pattern)
```

or

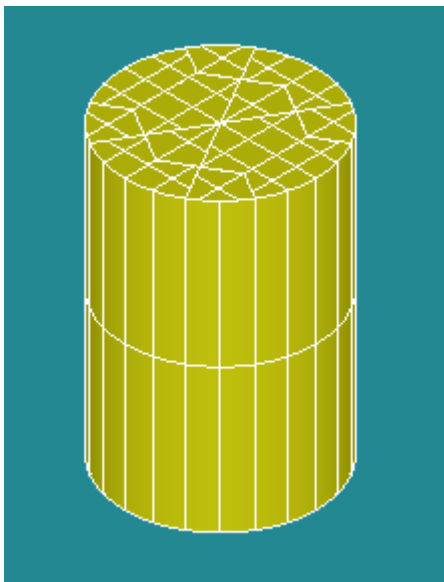
```
geompy.MakeDividedDiskPntVecR(Center,
                               Vector, Radius, Pattern)
```



Disk with a hexagonal pattern



Disk with a square pattern



Cylinder with a hexagonal pattern

Divided cylinder is a cylinder divided into **blocks** for easy hexahedral meshing.

To create a **Divided Cylinder** in the **Main Menu** select **New Entity** → **Advanced** → **Divided Cylinder**.

A divided cylinder can be defined base radius and height. The cylinder is divided by a predefined square or hexagonal pattern.

In TUI this operation is available through command

```
geompy.MakeDividedCylinder(R, H, Pattern).
```

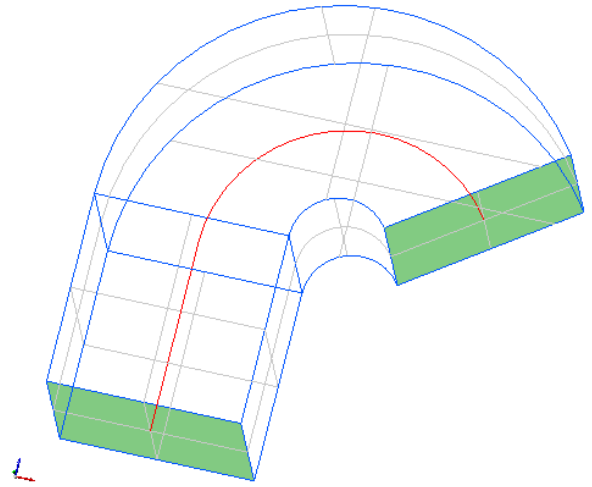

Restore Path

Restore Path algorithm takes at input a pipe-like shape (shell or solid) and its start and end faces (or wires) and creates a wire that represents a path made by gravity centers of consequent sections of the shape.

This operation is available from the main menu via **New Entity → Generation → Restore Path**.

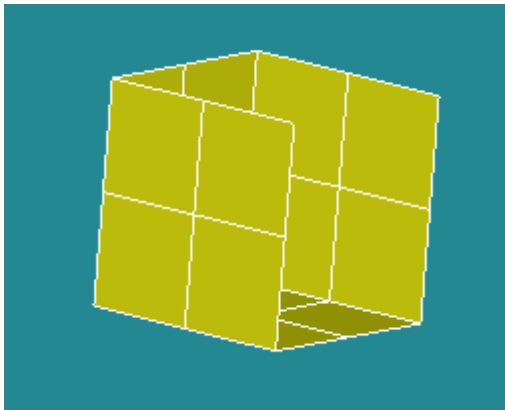
Alternatively it is possible to use Python command

```
geompy.RestorePath(shape, base1, base2).
```



The path between two green faces is shown in red

Boolean Operations on Groups



Result of Cut operation on two groups of faces of a box.

Boolean operations on groups: **Union**, **Intersect** and **Cut** – are now available in Geometry module.

These operations are available from the main menu via **New Entity → Groups** or using the corresponding Python functions:

- `UnionGroups(g1, g2);`
- `UnionListOfGroups(list);`
- `IntersectGroups(g1, g2);`
- `IntersectListOfGroups(list);`
- `cutGroups(g1, g2);`
- `cutListOfGroups(list1, list2).`

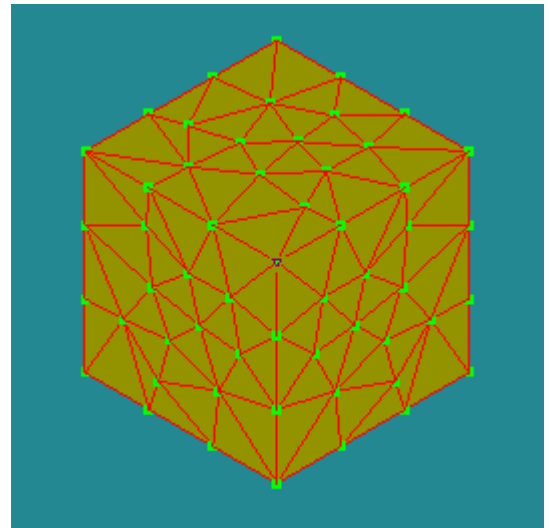
0D Elements on Element Nodes

It is now possible to create 0D elements on all nodes of the selected mesh, sub-mesh, or a group of mesh elements or nodes.

For this, select in the main menu **Modification** → **Add** → **0D Elements on Element Nodes**.

The target elements can be picked in the viewer, selected using filters or their IDs can be input directly in the corresponding field.

It is also possible to add all created 0D elements to a new or already existing group.

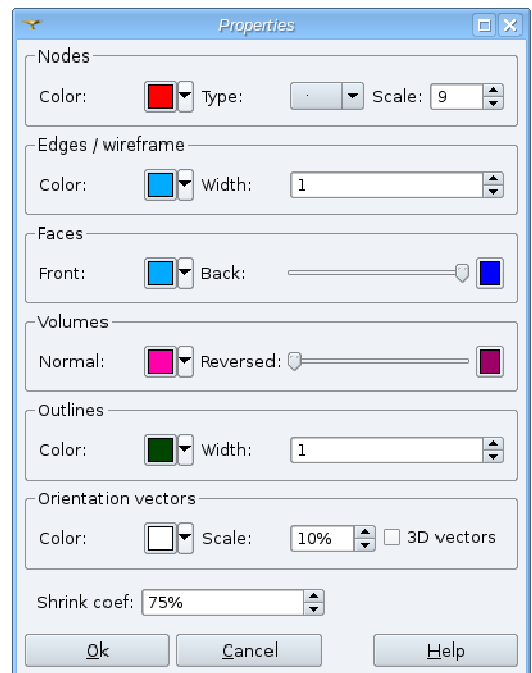


Mesh on box with created 0D elements (green)

Redesigned Mesh Properties dialog box

In Mesh module, the "Properties" dialog box has been redesigned to show only the properties of mesh entities that are present in the mesh (sub-mesh, group) object.

In addition, separate properties for colors of volume elements and element outlines width have been introduced.



HEXABLOCK MODULE

SALOME team is pleased to announce that HEXABLOCK module is available now for using. The HEXABLOCK module is designed to help the users to build hexahedral meshes. Please refer to the documentation of the HEXABLOCK module for more details.

MED MODULE CHANGE LOG

API Modification in Python only

- `python -i fileUsingMEDCouplingOrMEDLoader.py` will now read, if any, the file pointed to `PYTHONSTARTUP` → useful for completion.
- Python API of `MEDFileMesh.getTime: time, it, order` → `it, order, time` (like in functions `MEDLoader.GetAllFieldIterations`, `MEDFileFieldMultiTS.GetTimeSteps`).
- Python API of `MEDFileField1TS.getTime: time, it, order` → `it, order, time` (like in functions `MEDLoader.GetAllFieldIterations`, `MEDFileFieldMultiTS.GetTimeSteps`).
- Python API of `MEDFileField1TSwithoutDAS::setTime: time, it, order` → `it, order, time` (like in `MEDLoader.GetAllFieldIterations`, `MEDFileFieldMultiTS.GetTimeSteps`).
- C++/Python API `MEDFileMesh::setTime: time, it, order` → `it, order, time`.

API Modification in C++/Python

- Class name `MEDFieldField` → `MEDFileField`.
- `MEDCoupling[*]Mesh::renumberCells` - addition of default value (`true`) for check parameter.
- `MEDCoupling[PointSet, UMesh, UMeshDesc]::buildPartOfMySelf` - addition of default value (`true`) to for keepCoords parameter.
- `MEDCouplingFieldDiscretization::renumberCells` - addition of default value (`true`) for check parameter.
- `MEDCouplingFieldDouble::renumberCells` - addition of default value (`true`) for check parameter.
- `DataArray[Int, Double]::sort` – added `asc` boolean parameter for sorting in ascending/descending order.
- `DataArrayInt::getIdsEqualList`, `DataArrayInt::getIdsNotEqualList`.
- `MEDCouplingPointSet::findBoundaryNodes`.
- `MEDCouplingUMesh: findCellsIdsOnBoundary` → `findCellIdsOnBoundary`.

Behavior modification

- `MEDCouplingUMeshIntersect2DMeshes`: now part of cells in mesh `m1` not intersected by `m2` is returned; so the sum of the `abs(area)` of cells of returned mesh is equal to the `abs(area)` of cells of `m1`.
- `MEDFileUMesh::getFamilyFieldAtLevel`: does not throw exception if `meshDim==1` and there's no family field on it.
- `MEDFileMesh::setFamilyIdUnique`: throws exception if `id` is already used.
- `DataArrayInt::MakePartition`: added error protection when partitioning between with invalid groups (`ids` not in `[0, maxsize)` range) is made.
- `MEDCouplingUMesh::getMeasureField`: now area of 2D quadratic cells (`TRI6`, `QUAD8`, `QPOLY`) is computed correctly.

New features

- MEDFileUMesh::duplicateNodesOnM1Group : node duplication on dim-1 group of cells for cracking.
- MEDFileMesh::normalizeFamIdsTrio, MEDFileMesh::normalizeFamIdsMEDFile, MEDFileMesh::ensureDifferentFamIdsPerLevel for family numbers renumbering.
- MEDFileField1TS::setMeshName, MEDFileFieldMultiTS::setMeshName, MEDFileData::changeMeshName, MEDFileData::changeMeshNames.
- MEDFileData::unPolyze.
- Structures returned by MEDFileFieldMultiTS::getTimeStepAtPos and MEDFileFields::getFieldAtPos are now writable.
- MEDCouplingUMesh::unbutterfly2D using Jarvis march for convex hull.
- MEDCouplingUMesh::getSkin.
- MEDCouplingUMesh::partitionBySpreadZone.
- MEDCouplingUMesh::giveCellsWithType.
- MEDCouplingUMesh::simplifyPolyhedrons.
- MEDCouplingUMesh::computeFetchedNodeIds.
- MEDCouplingUMesh::computeNeighborsOfCellsAdv.
- MEDCouplingUMesh::ExtractFromIndexedArrays, MEDCouplingUMesh::RemoveIdsFromIndexedArrays
- MEDCouplingPointSet::duplicateNodesInCoords, MEDCouplingUMesh::duplicateNodes, MEDCouplingUMesh::duplicateNodesInConn.
- MEDCouplingUMesh::setPartOfMySelf, MEDCouplingUMesh::SetPartOfIndexedArrays, MEDCouplingUMesh::SetPartOfIndexedArraysSameIdx.
- MEDCouplingUMesh::AggregateSortedByTypeMeshesOnSameCoords.
- dataArrayInt::getIdsInRange.
- dataArrayInt::buildUnique.
- dataArrayInt::partitionByDifferentValues.
- MEDCouplingFieldDouble::buildSlice.
- Addition of geometric types SEG4, TRI7, QUAD9, HEXA27.
- MEDCouplingUMesh::writeVTK manages polyhedrons.
- nullifiedTinyCoeffInCrudeMatrixAbs, nullifiedTinyCoeffInCrudeMatrix, getMaxValueInCrudeMatrix for MEDCouplingRemapper.
- Version of MEDCoupling available by calling MEDCouplingVersionStr, MEDCouplingVersion or MEDCouplingVersionMajMinRel.
- True area computation of TRI6 QUAD8 and QPOLYG (in MEDCouplingUMesh::getMeasureField).
- Cartesian/Cartesian, Unstructured/Cartesian, Cartesian/Unstructured in MEDCouplingRemapper in POP0.
- CORBA API Addition of getName and getInfoOnComponents in idl in dataArrayServant and MEDCouplingFieldDoubleServant interfaces to retrieve information to improve performance in PARAVIS display using ParaMEDCorbaReader.
- CORBA API version of MED available for the client in case of evolution of the serialization format.

Python interface enhancement

- o `DataArrayInt.__contains__`.
- o `DataArrayInt.index` as Python lists.
- o `MEDCouplingUMesh.__setitem__`.
- o `__getitem__`, `__setitem__` in `MEDFileMeshes`.
- o `__getitem__`, `__setitem__` in `MEDFileFields`.
- o `__getitem__` in `MEDFileFieldMultiTS`.
- o Possibility of using constructors instead of static methods.
- o `MEDFileFieldMultiTS.New(fileName)`: read the first field as `MEDFileMesh` that reads the first mesh.
- o `__getitem__` in `MEDCouplingFieldDouble`.
- o `MEDFileFieldMultiTS.__delitem__` to suppress time steps
- o Iterator on `MEDFileFields` and `MEDFileFieldMultiTS` and `MEDFileMeshes`.
- o `DataArrayInt.getDifferentValues`.

Bugs solved

- o Bug correction in `MEDCouplingUMesh::buildDescendingConnec` on mesh with `meshDim==1`.
- o `MEDCouplingUMesh::checkCoherency` did not checked that `finishInsertingCells` has been called.
- o `DataArrayDouble::applyfunc`: check the number of components and throw an exception in case of invalid input instead of memory corruption.
- o Invalid `MEDCouplingUMesh::unPolyze` on PENTA6.
- o Bug in `DataArray::getInfoOnComponents`.
- o Bug in constructor of `MEDFileFieldMultiTS(fname, fieldName)` with MED file containing several fields with different number of components.
- o `MEDCouplingFieldDouble::getLocalizationOfDiscr` on fields on GAUSS points.

DOCUMENTATION IMPROVEMENTS

The SALOME documentation is now supplied with the top-level index file that presents the documentation by modules and by themes. This will help developers by giving them direct access to important topics (architecture, module development, etc...). Please refer to the documentation archive supplied with SALOME v6.6.0 Installation Wizard for more details.

OTHER IMPROVEMENTS

- BLSURF plug-in now supports 1D sub-meshes.
- The option to take units into account or not is now available at STEP import.
- Support of import and export of GMF format files used by DISTENE meshers has been introduced in Mesh module.
- It has become possible to synchronize an OCC view with a VTK view and vice versa.
- In YACS module, "Save schema with state" GUI function allows now saving in the same XML file both the graph itself and its current output.
- The information about version of each SALOME module is now stored in the study and can be retrieved, if necessary. In GUI it is available via "File / Properties" menu. In TUI it can be obtained using `salome.GetComponentVersion()` function.
- The name of the geometrical object selected in the viewer is now displayed in popup menu title (if only one object is selected).
- Popup menu items "Hide children" / "Show children" have been renamed into "Conceal child items" / "Disclose child items" correspondingly to avoid confusion with show/hide objects in the viewer.
- New item object browser popup menu "Show only children" allows erasing all shapes from the current viewer and displaying only children (sub-shapes) of the selected geometrical objects.
- New `geompy.RestoreShape(theStream)` method allows reading shapes from BRep binary stream.
- The preference, which allows switching on/off the display of "Color Legend" after creation of the presentation, has been added in ParaVis module.
- New TUI function `geompy.MakeVertexInsideFace(theFace)` creates an arbitrary vertex on the given face, strictly inside the face boundary (at a non-zero distance from the boundary).
- Default parameters have been provided for TUI functions `MakeFilling` and `MakeFillingNew`.
- NETGEN 1D-2D-3D meshing algorithm now allows usage of quadrangular 2D sub-meshes for generation of pyramids.
- Ascending order is now used for sorting in the Object browser by default (when sorting is switched on).
- Shading, Wireframe and Shading with Edges display mode options have become available for a shape in "Bring to front" state in OCC Viewer.
- In Mesh module, the "Create group" dialog box has been extended to allow creation of groups of 0D elements.
- Support of discrete elements of type Ball has been added to Mesh module. The following operations on a mesh of discrete elements are available:
 - Import from the MED file discrete elements of type MED_BALL that have "diameter" attribute.
 - Show discrete elements in the VTK viewer as squares.
 - Creation of groups of discrete elements by selecting elements either interactively in the viewer or using filters (ball-specific criterion "Diameter" has been introduced).
 - Export the whole mesh into MED file.
 - Make a compound mesh.
 - Copy a mesh/group.
 - Unite/intersect/cut groups.
 - Find element by point.
 - Add an element to the mesh.
 - Remove an element.
 - Translation, Rotation, Symmetry, Scale transformations.

- Two new methods have been added to python interface `geompy`:

```
GetAngleVectors(theShape1, theShape2, theFlag = True)
```

```
GetAngleRadiansVectors(theShape1, theShape2, theFlag = True)
```

These functions work only with vectors, taking into account their orientation. If `theFlag = False`, the biggest angle is returned, otherwise – the smallest one.



BUG CORRECTIONS

GUI MODULE (IAPP)

21670	<p><i>Summary:</i> EDF 2265 GUI: Update GUI documentation</p> <p>User's documentation for GUI module has been updated.</p>
21687	<p><i>Summary:</i> EDF 2306 STUDY: Dimension of the preference window</p> <p>The contents of "Preferences" dialog box has been made scrollable, to avoid an oversized dialog box on some desktop configurations.</p>
21704	<p><i>Summary:</i> [CEA 576] On windows version, the splash screen hides firewall dialog box</p> <p>Salome splash screen can now be closed by mouse click on it.</p>
21707	<p><i>Summary:</i> [CEA 585] Sort in object browser should be ascending by default</p> <p>Sorting in Object browser, when enabled, is now done in ascending order by default.</p>
21760	<p><i>Summary:</i> [CEA 605] adding a preference in SALOME</p> <p>New "Hide splash screen" preference allows disabling the splash screen.</p>
21761	<p><i>Summary:</i> [CEA 606] no effect for disableSplashScreen=true</p> <p>The user preferences files automatically written by the application now include a notice that warns the user that any changes manually made in these files can be lost.</p>
21769	<p><i>Summary:</i> EDF OCCT: Closing the OCC viewer leads to SIGSEGV (OCCT dev)</p> <p>A crash during clean-up of OpenGL resources has been eliminated.</p>
21779	<p><i>Summary:</i> [CEA 614] Ctrl +C with Salome Help</p> <p>Ctrl+C shortcut for quick copy is now usable in the built-in SALOME web browser.</p>
21781	<p><i>Summary:</i> [CEA 616] Suppress "." in the user's config files</p> <p>The user preference files are not saved as hidden (with "." prefix) as they are already stored in a hidden directory (~/.config/salome) on Linux.</p>

21804	<p><i>Summary:</i> [CEA 635] Name of the window when loading a study</p> <p>A complete basename of the study is now displayed in the SALOME desktop window title instead of a short basename.</p>
21808	<p><i>Summary:</i> [CEA 662] Synchronize OCC and VTK views</p> <p>It has become possible to synchronize an OCC view with a VTK view. This allows comparing the geometry with the mesh, for example, to understand why the mesh quality is bad at some places.</p>
21813	<p><i>Summary:</i> [CEA 642] Tool tips not always available in multi curve display in Plot2D view</p> <p>In Plot2D view tooltips are now shown for all curves (not only for the first one found by the iterator).</p>
21814	<p><i>Summary:</i> [CEA 643] wheel-zoom per axis in plot2D viewer</p> <p>In Plot2D view it has become possible to zoom independently different axes: V, V2 or H - by selecting the corresponding scale with the mouse pointer and rotating the mouse wheel.</p>
21862	<p><i>Summary:</i> [CEA 666] invalid tooltips in multi curve display in Plot2D view</p> <p>The problem with incorrect curve point detection when calculating tooltip for the point under mouse cursor has been fixed.</p>
21953	<p><i>Summary:</i> EDF GUI 2435 : The "desktop" section of the preference file may prevent the opening of SALOME main window</p> <p>'Minimized' desktop state has been explicitly prohibited in storing/restoring SALOME desktop geometry functionality.</p>

KERNEL MODULE

21478	<p><i>Summary:</i> EDF 2083 ALL: Write the version of Salome used to create the study in the HDF file</p> <p>The information about version of each SALOME module is now stored in the study and can be retrieved, if necessary. In GUI it is available via "File / Properties" menu. In TUI it can be obtained using <code>salome.GetComponentVersion()</code> function.</p>
21482	<p><i>Summary:</i> [CEA 539] inconsistency between <code>getAppliPath.py</code> and <code>runAppli</code></p> <p>The problem with incorrect processing of relative path to SALOME application, which caused failure of <code>runAppli</code> script, has been fixed.</p>

21688	<p><i>Summary:</i> [CEA 572] Issue when using <code>runSalome -k -t</code></p> <p>The problem with Salome shut down procedure, which caused a runtime error, has been fixed.</p>
22022	<p><i>Summary:</i> [CEA 723] Error in <code>killSalomeWithPort.py</code></p> <p>Fixed runtime error in <code>killSalomeWithPort</code> script caused by invalid processing of the <code>port omniORB</code> parameter. Additionally a problem that sometimes some servers were not killed by <code>killSalomeWithPort.py</code> has been fixed.</p>

GEOM MODULE

20014	<p><i>Summary:</i> EDF 833 GEOM : Regression from 3.2.9 with <code>GetInPlace</code></p> <p>Fixed regression (instability) in Partition algorithm.</p>
20952	<p><i>Summary:</i> EDF 1505 GEOM: The type of the geometrical primitives is lost by importing .step files</p> <p>Improved <code>KindOfShape</code> functionality to recognize some primitives, imported from STEP files, generated by SolidWorks application.</p>
21433	<p><i>Summary:</i> EDF GEOM SMESH: fail to mesh an ellipsoid using the new cartesian algo</p> <p>OCCT bug 22809 has been fixed in version 6.5.4 of Open CASCADE Technology. Temporary workaround in SALOME has been removed.</p>
21437	<p><i>Summary:</i> EDF 2021 GEOM: Problem of display of a shape</p> <p>Fixed problem of OCCT visualization for specific case of surfaces.</p>
21476	<p><i>Summary:</i> EDF 2082 GEOM: Minimize the number of C1 continuous edges in a wire</p> <p>It has become possible to fuse (unite) collinear (C1 at joint) edges of a wire using "Fuse collinear edges within a wire" operation.</p>
21477	<p><i>Summary:</i> EDF 1632 GEOM: Suppress a vertex from a wire</p> <p>The option "Fuse collinear edges to allow bigger radius" has been introduced in <code>Fillet1d</code> functionality for automatic fuse of collinear edges of working wire. This allows building fillets with bigger radius.</p>
21483	<p><i>Summary:</i> EDF 1638 GEOM: Retrieve non blocks solids and faces from a shape</p> <p>A new measurement "Get non blocks" returns all non-blocks solids and faces from the given shape.</p>

21511	<p><i>Summary:</i> EDF 2153 GEOM: Choose to take units into account or not when importing a STEP file</p> <p>The fix for this problem has been made in issue OCC22967.</p> <p>The option to take units into account or not is now available at STEP import.</p>
21525	<p><i>Summary:</i> EDF GEOM: Tolerance degradation issue and partition failure</p> <p>The problems with Boolean Cut operation have been fixed.</p>
21537	<p><i>Summary:</i> [CEA 554] Add behavior in GetAngle and GetAngleRadians</p> <p>A flag has been added to functions GetAngle and GetAngleRadians to define, which angle between two shapes is taken into account: the least if theFlag is True, and the greatest if theFlag is False.</p>
21538	<p><i>Summary:</i> EDF GEOM: GetShapesOnShapeAsCompound fails on cylinders for some cases</p> <p>A regression, which appeared after the implementation of a new functionality in OCCT 6.5.4., has been fixed.</p>
21546	<p><i>Summary:</i> EDF 2129 GEOM: Different behavior between circle and exploded base edge of a cylinder</p> <p>The fix for this problem has been made in issue OCC23174</p> <p>The algorithm finding surfaces now can check if the wire of a surface is closed in the parametric space of the found surface, and the surface is rejected if the wire is not closed. This allows avoiding non-planar base faces.</p>
21565	<p><i>Summary:</i> [CEA 557] Opposite of the pipe</p> <p>The fix for this problem has been made in issue OCC23367</p> <p>New Restore Path algorithm takes at input a pipe-like shape (shell or solid) and its start and end faces (or wires) and creates a wire that represents a path made by gravity centers of consequent sections of the initial shape.</p> <p>In python interface it is <code>geompy.RestorePath(shape, base1, base2)</code>.</p>
21568	<p><i>Summary:</i> EDF GEOM: PipeTShape with Fillet and prepared for hexa mesh cannot be built</p> <p>Fixed regression of PipeTShape creation algorithm (with fillet and partitioning).</p>
21573	<p><i>Summary:</i> EDF 2230 GEOM: Cut between 2 shapes crashes Salome</p> <p>The fix for this problem has been made in issue OCC23248</p> <p>The value of angular tolerance in surface intersection algorithm has been corrected.</p>

21671	<p><i>Summary:</i> EDF 1829 GEOM : Bring to front selected objects (continuation)</p> <p>Shading, Wireframe and Shading with Edges display modes have become available for a shape in "Bring to front" state in OCC Viewer.</p>
21672	<p><i>Summary:</i> [CEA 565] Dump Study from script</p> <p>Incorrect Python dump of a study created by the script doing certain steps on a group has been fixed.</p>
21675	<p><i>Summary:</i> [CEA] Not completed partition</p> <p>The fix for this problem has been made in issue OCC23218</p> <p>Search of the maximal value of the distance function has been replaced by Fibonacci search in the method calculating tolerance.</p>
21678	<p><i>Summary:</i> [CEA 567] import stp or wrong stp files</p> <p>The user is now warned if the imported STEP file contains no geometrical information.</p>
21683	<p><i>Summary:</i> EDF 678 GEOM: Partition failure</p> <p>The problem with indexing has been fixed in Partition algorithm</p>
21689	<p><i>Summary:</i> [CEA 571] Improve visualization of a selected object in GEOM</p> <p>The visualization of a selected object in Geometry module has been improved in the following ways:</p> <ul style="list-style-type: none"> ▪ The name of the geometrical object selected in the viewer is now displayed in popup menu title (If only one object is selected). ▪ Popup menu items "Hide children" / "Show children" have been renamed into "Conceal child items" / "Disclose child items" correspondingly to avoid confusion with show/hide objects in the viewer. ▪ New item object browser popup menu "Show only children" allows erasing all shapes from the current viewer and displaying only children (sub-shapes) of the selected geometrical objects.
21700	<p><i>Summary:</i> EDF 2316 GEOM: Regression in MakeFillet</p> <p>The fix for this problem has been made in issue OCC23252</p> <p>Some classes have been protected against adding faces to a shell that has already been added to a solid.</p>
21703	<p><i>Summary:</i> [CEA 577] Boolean operations on groups</p> <p>Boolean operations on groups usable from GUI and Python interface have been introduced in Geometry module.</p>

21706	<p><i>Summary:</i> [CEA 578] Python access to BrepTools::Read</p> <p>New <code>geompy.RestoreShape(theStream)</code> method allows reading shapes from BRep binary stream. Example of usage: <code>z = geom_shape.GetShapeStream() obj = geompy.RestoreShape(z).</code></p>
21710	<p><i>Summary:</i> [CEA 588] Increase the precision of the coordinates for the clipping</p> <p>The precision for the coordinates of a clipping plane has been made the same as for as for the length.</p>
21715	<p><i>Summary:</i> EDF 1661: Have default parameters for TUI function MakeFilling</p> <p>Default parameters have been provided for TUI functions <code>MakeFilling</code> and <code>MakeFillingNew</code>.</p>
21754	<p><i>Summary:</i> EDF 2278 GEOM : Problem with offset function and thickening = true on circular tubes</p> <p>The fix for this problem has been made in OCCT issue 23394 (version 6.5.4).</p> <p>The offset algorithm now takes into account the case of a planar face bounded by two edges that have circular curves.</p>
21755	<p><i>Summary:</i> EDF GEOM: Import of IGS file crashes SALOME</p> <p>A test on the nullity of the pointer <code>aUnintValue</code> has been added in <code>IGESImport.cxx</code> to avoid crash.</p>
21772	<p><i>Summary:</i> EDF 2336 GEOM: Non valid face created from two circles</p> <p>Fixing of invalid shapes, produced by shape building operations is now performed automatically.</p>
21777	<p><i>Summary:</i> [CEA 611] RemoveExtraEdges on a rounded cylinder</p> <p>The problem with bad shape resulted from <code>RemoveExtraEdges()</code> operation has been fixed in <code>BlockFix_SphereSpaceModifier.cxx</code></p>
21778	<p><i>Summary:</i> [CEA 612] MakeSolid on an unclosed shell in TUI</p> <p>A warning about unclosed shell has been provided in <code>geompy.MakeSolid</code> method. Check of arguments in Solid Construction GUI has been improved.</p>
21786	<p><i>Summary:</i> [CEA 620] AutoColor fails</p> <p><code>AutoColor</code> feature, that earlier worked only with groups, has been extended to also work with sub-shapes.</p>

21787	<p><i>Summary:</i> Deactivate preview for partition and Boolean operations in preferences</p> <p>For performance reasons, "Preview" check-box has been removed from dialog boxes of time-consuming operations: Fuse, Cut, Common, Section and Partition.</p>
21788	<p><i>Summary:</i> [CEA 622] preview on Boolean operations : no preview</p> <p>The preview of all Boolean operations has been corrected.</p>
21794	<p><i>Summary:</i> [CEA 626] Wrong common between a sphere and a box</p> <p>The fix for this problem has been made in issue OCC23374</p> <p>Procedure SameParameter is now applied to the auxiliary face to correct tolerance values of sub-shapes.</p>
21798	<p><i>Summary:</i> [CEA 627] Common incorrect between a box and an holed sphere</p> <p>The fix for this problem has been made in issue OCC23405</p> <p>Tolerance of intersection curves in 3D is now defined more precisely in case of plane/sphere intersection.</p>
21805	<p><i>Summary:</i> EDF GEOM: Internal name of STEP file is always used, name given by addToStudy is ignored</p> <p>The shape name, given by the user via <code>geompy.addToStudy</code> method is now correctly taken into account in case when the shape is read from a file.</p>
21809	<p><i>Summary:</i> [CEA 628] Impossible to create a solid from a shell created with <code>MakeShell</code></p> <p><code>MakeShell</code> functionality has been improved for a particular case of a single face with internal sub-shapes.</p>
21827	<p><i>Summary:</i> [CEA 650] TUI <code>setColor</code> on an object displayed in wireframe is ignored</p> <p>The shape's color is now properly taken into account by GEOM GUI Python API module (<code>LibGEOM_Swig</code>) even if this shape is displayed for the first time and the default display mode for GEOM GUI is "wireframe".</p>
21828	<p><i>Summary:</i> [CEA 651] TUI method to specify the number of Isos to show for an object</p> <p>New command <code>setIsos()</code> has been added to the Python API (SWIG interface) to allow changing the number of iso-lines from Python.</p>
21833	<p><i>Summary:</i> [CEA] problem with naming when exploding a step file</p> <p>Fixed regression caused by issue 0021805.</p>

21837	<p><i>Summary:</i> [CEA 654] Adding an option in MakeVertexOnSurface</p> <p>New TUI function <code>geompy.MakeVertexInsideFace(theFace)</code> creates an arbitrary vertex on the given face, strictly inside the face boundary (at a non-zero distance from the boundary).</p>
21838	<p><i>Summary:</i> [CEA 653] <code>createAndDisplayGO</code> needs user action to set color</p> <p>The flag <code>isUpdated</code> (update viewer) of <code>setColor()</code> function is now correctly taken into account.</p>
21846	<p><i>Summary:</i> EDF 2385 GEOM : Bug in the cut operation between a face and the result of a Fuse operation</p> <p>The fix for this problem has been made in issue OCC23431.</p> <p>Processing of closed edges has been improved in case of Boolean operations.</p>
21868	<p><i>Summary:</i> EDF GEOM : Bug with "What is"</p> <p>Exception raised from <code>whatIS</code> information on the b-spline curve consisting of two coinciding points (nodes or knots) has been fixed.</p>
21888	<p><i>Summary:</i> [CEA 710] Change orientation does not work on edges</p> <p>"Change Orientation" operation has been improved to reverse orientation of curves of edges and wires.</p>
21936	<p><i>Summary:</i> [CEA 698] GEO-GUI : Several methods to get the entry of one object, several different results</p> <p>An explanation about internal nature of mentioned methods has been added to the documentation.</p>
21937	<p><i>Summary:</i> [CEA 697] "structelem" modulus does not run if 2 objects have the same name in ObjectBrowser</p> <p>Problem with same-named objects has been temporarily solved by specifying full path object name instead of short one, i.e. its path.</p>

SMESH MODULE

20749	<p><i>Summary:</i> EDF 1291 SMESH : Create 2D Mesh from 3D improvement</p> <p>The problem with object browser update during creation of 2D boundary elements from 3D has been eliminated.</p>
-------	---

20982	<p><i>Summary:</i> EDF 1547 SMESH: Creation of non-conformal quadratic pyramids</p> <p>For elements having corner nodes out of geometry, the option "Medium nodes on geometry" = false is automatically applied; the user is then warned that there are such elements. This approach allows avoiding of distorted elements creation at making a quadratic mesh.</p>
21339	<p><i>Summary:</i> EDF 1928 SMESH: Create group of 0D elements</p> <p>A full set of operations on 0D elements has been implemented in GUI.</p> <ul style="list-style-type: none"> • Create/Edit Group (standalone group + group on geometry); • Filters (at least 7 filters: Belong to Geom, Belong to Plane, Belong to Cylinder, Belong to Surface, Lying on Geom, Range of IDs, Color of Group); • Union, Cut, Intersect Groups; • Group of Underlying Elements.
21374	<p><i>Summary:</i> EDF 1898 SMESH: Extrusion of a node to have an edge</p> <p>Extrusion of 1D elements has been improved to avoid creation of excess edges when extruding nodes.</p>
21382	<p><i>Summary:</i> EDF 1985 SMESH: Read/write of .mesh files (GMF format)</p> <p>It has become possible to import and export GMF format files (with extensions .mesh and .meshb) used by DISTENE meshers in Mesh module.</p>
21459	<p><i>Summary:</i> EDF 1495 SMESH: Manipulation of discrete elements with attributes</p> <p>Discrete elements of type MED_BALL with "diameter" attribute are now supported by SMESH module and can be displayed in VTK viewer as squares. The following operations on a mesh of discrete elements imported from a MED file are possible:</p> <ul style="list-style-type: none"> • Create groups of discrete elements by selecting them either interactively in the viewer or using filters (ball-specific criterion "Diameter"). • Make a compound mesh • Copy a mesh/group • Unite/intersect/cut groups • Find element by point • Add/Remove a mesh element <p>Translation, Rotation, Symmetry, Scale transformations</p>
21531	<p><i>Summary:</i> EDF 2198 SMESH: Improve the document "Integration of new meshing algorithm as plug-in to SALOME Mesh module"</p> <p>The user's guide "Integration of new meshing algorithm as plug-in to SALOME Mesh module" has been updated and in particular extended with the information on implementation of Python API, Dump Python and Connection to SALOME notebook.</p>
21542	<p><i>Summary:</i> EDF 1699 SMESH: Reorient a group of faces</p> <p>"Reorient faces by vector" operation, which allows changing orientation of a set of neighboring faces, has been added to Mesh module.</p>

21543	<p><i>Summary:</i> EDF 1978 SMESH: Viscous layer for 2D meshes</p> <p>Viscous layers 2D hypothesis is now available as an additional hypothesis for creation of viscous layers on geometrical faces.</p>
21559	<p><i>Summary:</i> EDF 2175 SMESH: Hexa/Tetra mixed meshes</p> <p>The problem with "Change sub-mesh priority" dialog box has been solved. Now when this dialog is invoked for the first time, concurrent sub-meshes are shown in the same order as they are computed by default.</p>
21669	<p><i>Summary:</i> EDF 2262 SMESH: "Mesh information" takes a long time on small mesh with joint elements</p> <p>"Mesh Information" dialog behavior has been improved to avoid performance problems if the mesh has a large number of groups and/or sub-meshes. In this case the information is shown partially and the user can browse it chunk by chunk.</p>
21680	<p><i>Summary:</i> EDF 2288 SMESH: creation of 0D elements from other elements</p> <p>It has become possible to create 0D elements on nodes of selected elements or nodes, where 0D elements are not yet present.</p>
21711	<p><i>Summary:</i> [CEA 579] Simplify Properties dialog accordingly to dimension of mesh</p> <p>"Properties" dialog box has been redesigned to show only the properties of mesh entities that are present in the mesh (sub-mesh, group) object. In addition, separate properties for colors of volume elements and outlines width have been introduced.</p>
21720	<p><i>Summary:</i> EDF SMESH: Body Fitting hypothesis dialog - Switching between the definition mode resets the previous values</p> <p>The input parameter values of Body Fitting meshing algorithm, are now preserved when switching from "Spacing" to "Coordinates" mode of grid definition and backwards.</p>
21723	<p><i>Summary:</i> EDF SMESH: Calculation of Aspect Ratio</p> <p>The Aspect Ratio computation algorithm for quadrilateral elements has been fixed to set Aspect Ratio of a square to 1.0.</p>
21758	<p><i>Summary:</i> [CEA] Internal test fails with Netgen3D</p> <p>A regression in Netgen 3D algorithm caused by implementation of MED_BALL elements has been eliminated.</p>
21759	<p><i>Summary:</i> EDF 1785 SMESH: A group of nodes created by filters is incomplete</p> <p>The group of nodes creation algorithm has been improved to eliminate the problem with missing elements.</p>

21762	<p><i>Summary:</i> EDF 2333 : Bug in composite side algorithm on partition result</p> <p>"Composite side discretization" algorithm has been improved to correctly define the continuity of wrongly oriented edges.</p>
21764	<p><i>Summary:</i> EDF SMESH: QUAD9 elements not counted in the mesh computation summary</p> <p>The dialog shown after mesh computation has been updated to give the number of elements of each available type.</p>
21765	<p><i>Summary:</i> EDF 2334 SMESH: ConvertToQuadratic with medium node on geometry generates bad elements</p> <p>Wrong positioning of medium nodes by ConvertToQuadratic() in a mesh computed by GHS3D mesher has been fixed.</p>
21783	<p><i>Summary:</i> EDF 785 SMESH : Bad display of Groups of Volumes</p> <p>Regression in visualization of groups produced by implementation of MED_BALL has been eliminated.</p>
21797	<p><i>Summary:</i> EDF 2343 SMESH: Evaluation of meshes give very different results compared to real results</p> <p>The problem with high difference between meshes evaluated and computed with use of NETGEN 1D-2D-3D and BLSURF meshers has been fixed.</p>
21799	<p><i>Summary:</i> EDF 2344 SMESH: Existing groups are not taken into account when adding new elements into groups</p> <p>The problem with incorrect addition of new elements to already existing groups has been fixed.</p>
21800	<p><i>Summary:</i> EDF 2346 : Bug in the dump of ExtrusionAlongpathX</p> <p>An error in python dump has been fixed.</p>
21801	<p><i>Summary:</i> EDF 2143 SMESH: random order of groups returned by ExtrusionSweepObject2D</p> <p>Extrusion and Revolution operations of Mesh module have been improved to return items of new groups arranged in the same order as in the original groups.</p>
21802	<p><i>Summary:</i> EDF 2350 SMESH OCC: Crash when editing Nb Segments hypothesis</p> <p>The fix for this problem has been made in issue OCC23403</p> <p>The crash caused by double freeing of memory when a lexical error is detected by the parser is now avoided.</p>
21806	<p><i>Summary:</i> [CEA 637] Wrap error message on Mesh Computation Failed Dialog</p> <p>Ergonomics of "Mesh computation results" dialog box has been slightly improved:</p>

	<ul style="list-style-type: none"> • long error & warnings messages wrapped to several lines • buttons "Show sub-shape", "Publish sub-shape" moved to the bottom part of dialog box • initial size of the dialog box slightly increased.
21815	<p><i>Summary:</i> EDF 2347 SMESH : Error when meshing vertices</p> <p>Single nodes are now displayed correctly by clicking on the eye button.</p>
21821	<p><i>Summary:</i> EDF 2356 SMESH: Wrong GHS3D mesh with Viscous Layer hypothesis</p> <p>Mesh generation on a circular edge using Viscous layers hypothesis has been improved.</p>
21823	<p><i>Summary:</i> [CEA 649] Compute failed without any errors</p> <p>A warning is issued if an algorithm reports no errors but no elements are assigned to some sub-shapes.</p>
21824	<p><i>Summary:</i> EDF 2364 SMESH : Unknown exception when dumping a Study</p> <p>A problem with Python Dump has been fixed.</p>
21825	<p><i>Summary:</i> Error in the example of "Projection Algorithms" in the user's guide</p> <p>A problem with Projection 3D meshing algorithm has been fixed.</p>
21826	<p><i>Summary:</i> EDF 2367 SMESH : Hexaedron failure</p> <p>Regular 1D meshing algorithm now correctly uses the indicated Number of Segments.</p>
21845	<p><i>Summary:</i> EDF 1855 SMESH : A no-regression test fails on Radia1Quadrangle meshing</p> <p>A regression in Radia1Quadrangle algorithm has been fixed.</p>
21861	<p><i>Summary:</i> EDF 2226 : Documentation of option in split quadrangles is unclear</p> <p>New option 'Minimum diagonal' has been added as default to the "Cutting of quadrangles" dialog box. It allows splitting the quadrangle by its minimum diagonal. Documentation about Cutting Quadrangles operation has also been corrected.</p>
21869	<p><i>Summary:</i> [CEA 672] Clipping always apply even with auto-apply not checked</p> <p>The problem with auto-applying clipping parameters in edition mode has been fixed.</p>
21882	<p><i>Summary:</i> [CEA 673] Add preference for default color for mesh groups</p> <p>The color for mesh group preference has been added to have a different color than the default color of surface mesh.</p>

21891	<p><i>Summary:</i> EDF 2398 : Error when dumping a study with non-historical mode</p> <p>SMESH python-dump now works correctly if a hypothesis is removed after mesh computation.</p>
21897	<p><i>Summary:</i> EDF 1495 SMESH : Problem with groups when building a compound with MED-Balls</p> <p>"Build Compound mesh" function now works correctly in case of a large number of meshes and groups.</p>
21905	<p><i>Summary:</i> EDF SMESH: Impossible to add a face without layers with "viscous layers"</p> <p>Minimal allowed size of "Sub-shapes preview chunk" preference has been set to 1 to avoid problems with face selection.</p>
21906	<p><i>Summary:</i> [CEA 579] Missing section in Mesh Properties Dialog</p> <p>"Wireframe" properties widget group has been added in "Properties" dialog box for a group of faces / volumes.</p>
21915	<p><i>Summary:</i> [CEA 685] Remove ASCII from the STL file selection filter in the import STL file dialog box</p> <p>The limitation to ASCII format in the file selection filter has been removed as import of STL files in binary format is also possible. Localization has also been added in STL read/write driver.</p>
21921	<p><i>Summary:</i> [CEA 690] GetVolume applied on many elements has performance issue</p> <p>Performance of python functions returning values of numeric functors for an element (such as GetLength(), GetArea(), etc.) has been improved.</p>
21924	<p><i>Summary:</i> EDF 2461 SMESH : Performance issue for mesh visualization</p> <p>The problem with increased memory consumption in case of display of coinciding 3D mesh elements has been fixed.</p>
21940	<p><i>Summary:</i> EDF 2460 SMESH : Issue with ConvertToQuadratic</p> <p>A 2D mesh on a periodic face without seam edges, but with a degenerated edge is now correctly converted to quadratic.</p>
21942	<p><i>Summary:</i> [CEA 700] Behavior of Mesh.Triangle(algo=smesh.NETGEN) command</p> <p>Python console now outputs a warning about a meshing algorithm ignored during mesh generation because there is an upper dimension algorithm generating the mesh of a lower dimension.</p>
21954	<p><i>Summary:</i> [CEA 706] Error at the mesh object creation</p> <p>It has become possible to create a mesh on a shape including empty compounds.</p>

21979	<p><i>Summary:</i> [CEA 709] Import/export GMF should not take into account the locale to write/read numbers</p> <p>Locale behavior at import/export of GMF files has been fixed.</p>
21980	<p><i>Summary:</i> [CEA 708] Import/export GMF does work only if the file extension is ".mesh"</p> <p>The user is notified about error at the attempt to load a GMF file with a wrong extension (not ".mesh" or ".meshb").</p>
21982	<p><i>Summary:</i> [CEA 713] Wrong 2D Projection</p> <p>Projection 2D meshing algorithm has been enabled to work with faces with multiple holes.</p>
21985	<p><i>Summary:</i> [CEA 715] "Clear mesh data" on a skin mesh without geometry does not preserve the skin mesh</p> <p>Behavior of "Clear mesh data" command has been changed for the cases when the mesh is not computed on the geometry (imported, compound or made from scratch). In these cases only the elements and nodes computed by algorithms are removed.</p>
21988	<p><i>Summary:</i> EDF 2484 SMESH : NETGEN and GDHS3D/BLSURF mixed meshes</p> <p>Fix BLSURF plug-in to create a conformal mesh in case if nodes on vertices are already computed</p>
21993	<p><i>Summary:</i> EDF SMESH : Crash during extrusion of a mesh along a path</p> <p>Crash during Extrusion operation is now avoided in case when the 1D mesh/sub-mesh path is computed using "Composite side discretization" algorithm.</p>
21994	<p><i>Summary:</i> [CEA 718] Wrong UseExisting1DElements and UseExisting2DElements documentation</p> <p>TUI documentation for UseExistingElements option has been corrected.</p>
21996	<p><i>Summary:</i> EDF SMESH 1549: Regression in projection algorithms</p> <p>Regression in projection algorithms has been fixed.</p>
21998	<p><i>Summary:</i> EDF 2496 SMESH : Paravis is opened when Show Only is invoked in SMESH</p> <p>The problem with common operations that caused unnecessary modules loading during "Show Only" / "Erase all" operations, has been fixed.</p>

MED MODULE

21673	<p><i>Summary:</i> [CEA 566] Bug in Sauvwriter when writing meshes having no family ids on nodes.</p> <p>A critical error in Sauvwriter in case of missing family IDs of nodes has been fixed.</p>
21742	<p><i>Summary:</i> [CEA 597] Missing Group on all cells after loading a sauv file</p> <p>SauvReader now avoids using as mesh name the name of the group created on all cells.</p>
21745	<p><i>Summary:</i> [CEA 600] Invalid load of SAUV file with mesh containing SEG3</p> <p>SauvReader has been fixed to convert connectivity of SEG3 elements according to MED convention while reading 1D meshes.</p>
21749	<p><i>Summary:</i> [CEA 601] Some missing groups in mesh after reading a SAUV file with SauvReader</p> <p>SauvReader now assigns different names to created equal groups. This fix has been implemented to avoid missing groups.</p>
21763	<p><i>Summary:</i> [CEA 607] Memory fault during reading of a sauv file</p> <p>SauvReader has been fixed to correctly process self-intersecting sub-meshes in a SAUV file.</p>
21771	<p><i>Summary:</i> [CEA 604] Segmentation fault while reading a sauv file</p> <p>SauvReader has been improved to correctly read SAUV files containing elements with different dimensions.</p>
21773	<p><i>Summary:</i> [CEA 608] Conversion of a med file to sauv file using MEDLoader</p> <p>Sauvwriter has been fixed to correctly write quadratic elements.</p>

YACS MODULE

21426	<p><i>Summary:</i> [CEA 541] Output of an YACS graph at the end of the execution</p> <p>New "Save schema with state" GUI function allows saving in the same XML file both the graph itself and its current output.</p>
-------	--

PARAVIS MODULE

21712	<p><i>Summary:</i> [CEA 581] Preference to display legend by default</p> <p>The preference, which allows switching on/off the display of "Color Legend" after creation of the presentation, has been added in ParaVis module.</p>
21721	<p><i>Summary:</i> [CEA 590] Invalid groups on cells display in ParaVis</p> <p>The algorithm determining entity of the family has been improved.</p>
21724	<p><i>Summary:</i> [CEA 592] Unable to load a MED file in paraview supplied with SALOME</p> <p>PV_PLUGIN_PATH environment variable has been redefined in the installation script for PARAVIS module to correctly open MED files.</p>
21725	<p><i>Summary:</i> [CEA 593] Rendering issue of a polyhedron</p> <p>The problem with reading polyhedrons from MED files has been fixed in MedReaderPlugin.</p>
21729	<p><i>Summary:</i> [CEA 595] Slow polyhedrons rendering in ParaVis</p> <p>The problem with slow rendering of polygons and polyhedrons has been fixed.</p>
21730	<p><i>Summary:</i> [CEA 596] Slice of polyhedron in PARAVIS returns no cell</p> <p>The problem caused by the overriding of the standard vtkCutter filter by the vtkEDFCutter class implemented in SALOME GUI module using the vtkObjectFactory mechanism has been fixed.</p>
21744	<p><i>Summary:</i> [EDF 2327] PARAVIS: ELNO Surface filter leads to crash of Salome</p> <p>Salome crash at application of the ELNO Surface filter is now avoided.</p>
21790	<p><i>Summary:</i> [CEA 613] No + in animation View</p> <p>ShowEvent is sent to internal widget to allow creating more than one animation view.</p>
21807	<p><i>Summary:</i> [CEA 641] ParaVis state not saved in the study</p> <p>The problem of saving paraview state (*.pvsm format) has been fixed.</p>
21944	<p><i>Summary:</i> [CEA 702] med2vtk tool fails with field records with more than 4 components</p> <p>The problem with med2vtk tool that caused aborting of MED file processing after the first detected problem (e.g. if the converted field was not compatible with VTK). Now med2vtk tries to process all fields in a med file ignoring minor errors.</p>

21990	<p><i>Summary:</i> EDF 2481: Importing a MED file freezes the application</p> <p>The application has ceased freezing in case of raising any SIGNAL (e.g. SIGSEGV) from Paraview code.</p>
-------	---

BLSURFPLUGIN MODULE

21381	<p><i>Summary:</i> EDF 1984 BLSURFPLUGIN: Sub-mesh with BLSURF</p> <p>BLSURF plug-in has been improved to support 1D sub-meshes.</p>
21785	<p><i>Summary:</i> [CEA 619] impossible to add an enforced vertex</p> <p>BLSURFPluginGUI_HypothesisCreator.cxx has been modified to correctly add the coordinates of enforced vertex.</p>
21811	<p><i>Summary:</i> EDF BLSURFPLUGIN GHS3DPLUGIN: If dump is not historical, dump of hypothesis is incomplete</p> <p>Python dump has been corrected to correctly dump some methods that do not change the value of a meshing parameter but add one more value.</p>
21812	<p><i>Summary:</i> [CEA 638] Error when using Use Internal Vertices on all faces</p> <p>"Use internal vertices on all faces" option of "BLSURF_Parameters" hypothesis has been corrected.</p>
21816	<p><i>Summary:</i> [CEA 644] Use internal vertices of all faces is not saved in the study</p> <p>"Use internal vertices of all faces" option is now correctly saved in the study.</p>
21818	<p><i>Summary:</i> [CEA 645] Error when using Use Internal Vertices</p> <p>A bug of incorrect treatment of "Use internal vertices on all faces" option of "BLSURF_Parameters" hypothesis has been fixed.</p>
21819	<p><i>Summary:</i> [CEA 646] BLSURF creates duplicate nodes, mesh impossible with GHS3D</p> <p>It has become possible to use GHS3D mesher on a 2D mesh generated by BLSURF with activated option "Use internal vertices on all faces".</p>
21988	<p><i>Summary:</i> EDF 2484 SMESH : NETGEN and GDHS3D/BLSURF mixed meshes</p> <p>BLSURF plug-in has been fixed to create a conformal mesh if nodes on vertices are already computed.</p>

GHS3PLUGIN MODULE

21775	<p><i>Summary:</i> [CEA 609] Errors in python API of SetEnforcedMesh in GHS3DpluginDC.py</p> <p>Conditions for size and group_name in SetEnforcedMesh have been reversed.</p>
21776	<p><i>Summary:</i> [CEA 610] SetEnforcedMeshwithGroup fails or succeeds depending in how we use it</p> <p>Usage of SetEnforcedMeshwithGroup() has been corrected to avoid critical error.</p>
21782	<p><i>Summary:</i> [CEA 617] Enforced vertices without geometry fails</p> <p>Regression in SetEnforcedMeshwithGroup() has been fixed.</p>
21843	<p><i>Summary:</i> [CEA 658] Segmentation fault when using GHS3D on a 2D group</p> <p>Crash of SALOME if a sub-mesh is created on a COMPOUND sub-shape, which is not a group, is now avoided.</p>
21983	<p><i>Summary:</i> [CEA 714] Imposed mesh not taken into account after HDF reload</p> <p>Persistence mechanism of GHS3Dplugin_Hypothesis has been fixed to correctly restore enforced meshes from the HDF study file.</p>

HEXOTICPLUGIN MODULE

21947	<p><i>Summary:</i> EDF 1987 : regression with Hexotic, holes are meshed</p> <p>A regression in Hexotic meshing plugin causing improper meshing of holes in a shape has been fixed.</p>
-------	--

NETGENPLUGIN MODULE

21676	<p><i>Summary:</i> EDF 2283 NETGENPLUGIN: Improve Netgen 1D-2D-3D to generate pyramids in case where input 2D mesh includes quadrangles</p> <p>NETGEN 1D-2D-3D meshing algorithm now allows usage of quadrangular 2D sub-meshes for generation of pyramids.</p>
21681	<p><i>Summary:</i> EDF 2246 NETGENPLUGIN: Local size not fully taken into account</p> <p>The local size on internal edges is now properly taken into account in NETGEN plug-in.</p>
21884	<p><i>Summary:</i> [CEA 679] SALOME crashes when using Netgen 2D without 1D algorithm</p> <p>A regression from implementation of non-conformal quadratic pyramids has been fixed.</p>

INSTALLATION PROCEDURE

21726	<p><i>Summary:</i> [CEA 594] Valgrind error with Python delivered in Salome package</p> <p>Python binaries are rebuilt with --without-pymalloc option and repackaged for all Linux platforms.</p>
-------	---

OCCT 6.5.4 BUG CORRECTIONS

This chapter lists bug corrections and improvements included in Open CASCADE Technology 6.5.4.

Note that only the issues connected with SALOME platform are listed below. Please, refer to Open CASCADE Technology version 6.5.4 Release Notes for a complete list of changes.

OCC22809	<p><i>Summary:</i> BRepIntCS does not find intersections of an ellipsoid and a line passing through its apex (and near it)</p> <p>Referenced by 0021433: EDF GEOM SMESH: fail to mesh an ellipsoid using the new Cartesian algorithm</p>
OCC22884	<p><i>Summary:</i> The attached face cannot be displayed in shading mode</p> <p>Referenced by 0021437: EDF 2021 GEOM: Problem of display of a shape</p>
OCC23100	<p><i>Summary:</i> Wrong adjustment of p-curves on a cylinder</p> <p>Referenced by 0021525: EDF GEOM: Tolerance degradation issue and partition failure</p>
OCC23129	<p><i>Summary:</i> BRepTools::OuterShell() works wrong - it always returns the first shell</p> <p>Referenced by 0021572: EDF 2229 GHS3DPLUGIN: Hole in a flattened hollowed sphere is meshed by ghs3d</p>
OCC23139	<p><i>Summary:</i> BRepFilletAPI_MakeFillet algorithm builds edges with wrong first and last parameters: out of p-curve</p> <p>Referenced by 0021550: EDF 1572 GEOM : Regression, a test script fails in a cut operation</p>
OCC23158	<p><i>Summary:</i> ApproxInt_PrmPrmSvSurfaces raises FPE (division by zero) signal</p> <p>Referenced by 0021568: EDF GEOM: PipeTShape with Fillet and prepared for hexa mesh cannot be built</p>
OCC23160	<p><i>Summary:</i> Cut operation with the attached shapes produces a not correct result.</p> <p>Referenced by 0021550: EDF 1572 GEOM: Regression, a test script fails in a cut operation</p>
OCC23162	<p><i>Summary:</i> BRepOffsetAPI_MakePipeShell works wrong</p> <p>Referenced by 0021563: EDF GEOM: Extrusion with scale factor sometimes gives wrong shapes</p>

OCC23174	<p><i>Summary:</i> BrepLib_MakeFace(wire) creates an invalid face on a wire of cylinder bottom</p> <p>Referenced by 0021546 EDF 2129 GEOM: Different behavior btw circle and exploded base edge of a cylinder</p>
OCC23196	<p><i>Summary:</i> Porting to the latest version of ftgl library</p> <p>According to the decision of SALOME CoTech, the latest ftgl (2.1.3rc5) is used with OCCT - in order to use the native ftgl on recent distributions of Linux platform.</p>
OCC23201	<p><i>Summary:</i> Projection algorithm produces wrong results.</p> <p>Referenced by 0021547: EDF 2194 GEOM : The partition btw an ellipsoid and a box gives a bad result</p>
OCC23218	<p><i>Summary:</i> Extra vertex produced by Boolean section in case of closed intersection curves.</p> <p>Referenced by 0021657: [CEA] Not completed partition</p>
OCC23248	<p><i>Summary:</i> Wrong result done by solid classifier algorithm for infinite point</p> <p>Referenced by 0021573: EDF 2230 GEOM: Cut between 2 shapes crashes Salome</p>
OCC23252	<p><i>Summary:</i> Fillet regression</p> <p>Referenced by 0021700: EDF 2316 GEOM: Regression in MakeFillet</p>
OCC23341	<p><i>Summary:</i> Wrong result done by 2D classifier algorithm for a point and a face.</p> <p>Referenced by 0021570: EDF 2166 GEOM: Boolean operation fail</p>
OCC23367	<p><i>Summary:</i> New functionality restoring the middle path of pipe-like shape</p> <p>Referenced by 0021565: [CEA 557] Opposite of the pipe</p>
OCC23374	<p><i>Summary:</i> BOP Common between a sphere and a box gives wrong result</p> <p>Referenced by 0021794 [CEA 626] Wrong common between a sphere and a box</p>
OCC23394	<p><i>Summary:</i> Problem with BrepOffset_MakeOffset on a cylindrical face</p> <p>Referenced by 0021754 EDF 2278 GEOM : Problem with offset function and thickening = true on circular tubes</p>

OCC23403	<p><i>Summary:</i> Crash when parsing an expression with lexical error</p> <p>Referenced by 0021802 EDF 2350 SMESH OCC: Crash when editing Nb Segments hypothesis</p>
OCC23405	<p><i>Summary:</i> BOP common produces one face instead of a solid</p> <p>Referenced by 0021798 [CEA 627] Common incorrect between a box and an holed sphere</p>
OCC23431	<p><i>Summary:</i> BOP Cut produces invalid shape</p> <p>Referenced by 0021846 EDF 2385 GEOM : Bug in the cut operation between a face and the result of a Fuse operation</p>
OCC23442	<p><i>Summary:</i> Provide the access to the functionality of <code>Geom2dHatch_Hacher</code> via <code>IntTools_Context</code></p> <p>Referenced by 0021538: EDF GEOM: <code>GetShapesOnShapeAsCompound</code> fails on cylinders for some cases</p>
OCC23464	<p><i>Summary:</i> Projection algorithm produces wrong results.</p> <p>Referenced by 0021547: EDF 2194 GEOM : The partition btw an ellipsoid and a box gives a bad result</p>
OCC23470	<p><i>Summary:</i> Boolean Fuse between two edges fails</p> <p>Referenced by 0021547: EDF 2194 GEOM : The partition btw an ellipsoid and a box gives a bad result</p>



SUPPORTED LINUX DISTRIBUTIONS AND PRE-REQUISITES

SALOME 6.6.0 supports Linux Debian 4.0 Etch 32bits and 64bits, Debian 5.0 Lenny 64bits, Debian Squeeze 6.0 64bits, Mandriva 2008 32bits and 64bits, Mandriva 2010 32bits and 64bits, Red Hat Enterprise 4.0 64bits, Scientific Linux 5.1 64bits, CentOS 5.5 64 bits and Windows XP 32bits. **SALOME 6.6.0** version has been mainly tested with below listed pre-requisites on Mandriva 2010 32bits and Debian 6.0 Squeeze 64bits platforms.

SALOME 6.6.0 comes with the same prerequisites versions on all supported platforms (with some exceptions). The table below lists the versions of the products used by SALOME platform. Other versions of the products can also work but it is not guaranteed.

Product	Version	GUI (IAPP)	KERNEL	GEOM	SMESH	VISU	MED	YACS	PARAVIS	HOMARD	HEXABLOCK	NETGENPLUGIN	GHS3DPLUGIN	GHS3DPRPLPLUGIN	BLSURFPLUGIN	HexoticPLUGIN	HEXABLOCKPLUGIN
gcc*	4.2**	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X
automake*	1.9**	X	X	X	X	X	X	X		X	X	X	X				X
autoconf*	2.59**	X	X	X	X	X	X	X		X	X	X	X	X	X	X	X
libtool*	1.5.6**	X	X	X	X	X	X	X		X	X	X	X	X	X	X	X
GNU make*	3.80**	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X
cmake	2.8.7								X								
Python	2.6.6	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X
Qt	4.6.3	X		X	X	X	X	X	X	X	X	X	X	X	X	X	X
Sip	4.13.2	X			X												
PyQt	4.9.1	X			X												
Boost	1.49.0	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X
Swig	1.3.40	X	X	X	X	X	X	X	X	X		X	X	X	X	X	X
OCCT	6.5.4	X		X	X	X	X		X	X	X	X	X	X	X	X	X
Qwt	5.2.1	X			X												
QScintilla	2.6.1							X									
OmniORB	4.1.6	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X
OmniORBpy	3.6	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X
omniNotify	2.1		X														
Hdf5	1.8.8	X	X	X	X	X	X		X	X		X	X	X	X	X	X
Med	3.0.6				X	X	X		X	X		X		X			
Vtk	5.8.0	X		X	X	X	X		X		X	X	X	X	X	X	X
numpy	1.5.1		X														
lapack	3.3.0		X														
graphviz	2.28.0	X	X	X	X	X	X	X				X	X	X	X	X	
Doxygen	1.8.0	X	X	X	X	X	X	X				X	X	X	X	X	X
NETGEN	4.9.13											X					
docutils	0.8.1	X	X	X	X	X	X	X		X	X	X	X	X	X	X	
metis	4.0						X										
scotch	5.1.11						X										
libxml2	2.7.8	X	X		X		X	X	X		X		X				
Distene MeshGems	1.0.1												X	X	X		
Hexotic	1.0														X	X	
sphinx	1.1.3		X	X	X			X		X	X						
expat	2.0.1							X									
libBatch	1.6.0		X														
cgns	3.1.3				X												
ParaView	3.14.0								X								
Homard	10.5									X							

*) Not included into SALOME Installation procedure
 **) Minimal required version

Product	Version	RANDOMIZER	SIERPINSKY	PYCALCULATOR	COMPONENT	CALCULATOR	HELLO	PYHELLO	LIGHT	PYLIGHT	ATOMIC	ATOMGEN	ATOMSOLV	HXX2SALOME	YACSGEN	JOBMANAGER
gcc*	4.2**	X	X	X	X	X	X	X	X	X	X	X	X	X		X
automake*	1.9**	X	X	X	X	X	X	X	X	X	X	X	X	X		X
autoconf*	2.59**	X	X	X	X	X	X	X	X	X	X	X	X	X		X
libtool*	1.5.6**	X	X	X	X	X	X	X	X	X	X	X	X	X		X
GNU make*	3.80**	X	X	X	X	X	X	X	X	X	X	X	X	X		X
Python	2.6.6	X	X	X	X	X	X	X	X	X	X	X	X		X	X
Qt	4.6.3		X			X	X	X	X		X	X	X	X		X
Sip	4.13.2											X				
PyQt	4.9.1								X		X					
Boost	1.49.0		X			X	X						X			X
Swig	1.3.40		X		X	X										
OCCT	6.5.4		X		X	X	X		X		X		X			
Qwt	5.2.1															
OmniORB	4.1.6	X	X	X	X	X	X	X				X	X			X
OmniORBpy	3.6	X	X	X	X	X	X	X				X	X			X
Hdf5	1.8.8		X		X	X			X		X					
Med	3.0.6		X	X	X	X										
Vtk	5.8.0		X						X	X	X		X			
graphviz	2.28.0	X	X	X	X		X	X			X					
Doxygen	1.8.0	X	X	X	X		X	X			X					
sphinx	1.1.3															X

*) Not included into SALOME Installation procedure

***) Minimal required version

The following products are not used in SALOME directly; they are only required to build other pre-requisite products.

Product	Version	Required by	Comment
tcl	8.5.8	Open CASCADE Technology, VTK	Optional
tk	8.5.8	Open CASCADE Technology, VTK	Optional
tclX	8.4.0	Open CASCADE Technology	Optional
jinja	2.6	Sphinx	
pygments	1.5	Sphinx	
setuptools	0.6c11	Sphinx	
freetype	2.4.10	Open CASCADE Technology	
ftgl	2.1.3	Open CASCADE Technology	
freeimage	3.14.1	Open CASCADE Technology	Optional
gl2ps	1.3.5	Open CASCADE Technology	Optional
Intel TBB	3.0	Open CASCADE Technology	Optional
xdata	0.9.6		Can be used to create 3 rd - party SALOME modules

NOTE: For some platforms SALOME uses prerequisites with patches (to fix different problems, like it is done in RPM) and defines specific configuration/compilation options. If you compile products without the Install Wizard we strongly recommend you to check configuration/compilation options using shell scripts located in config_files folder of the SALOME Installation Wizard.

SALOME 6.6.0 depends on a number of products for run time execution, others are necessary only for compilation or generation of development documentation (like doxygen for example). Below there is a list of mandatory and optional products.

Product	Compilation and Development		Execution		Remarks
	Mandatory	Optional	Mandatory	Optional	
gcc	X		X		
Automake	X				Except for PARAVIS
Autoconf	X				Except for PARAVIS
libtool	X				Except for PARAVIS
GNU make	X				
cmake	X				For PARAVIS only
Python	X		X		
Qt	X		X		
sip	X				
PyQt	X		X		
Boost	X		X		
Swig	X				
OCCT	X		X		
Qwt	X		X		
QScintilla		X		X	For YACS only Required only if used at compilation step
omniORB	X		X		
omniORBpy	X				
omniNotify	X		X		
Hdf	X		X		
Med	X		X		
Vtk	X		X		
numpy + lapack		X			
graphviz	X		X		In run-time required for YACS only
Doxygen	X				
NETGEN	X		X		For NETGENPLUGIN only
docutils		X			
cppunit		X			Used for unitary testing
mpi		X		X	Required only if used at compilation step
openpbs		X		X	Required only if used at compilation step
Lsf		X		X	Required only if used at compilation step
metis		X		X	Required only if used at compilation step
scotch		X		X	Required only if used at compilation step
libxml2	X		X		
Blsurf*	X	X	X	X	mandatory for BLSURF plugin, optional for Hexotic plugin
TetMesh-GHS3D*	X		X		For GHS3D and GHS3DPRL plugins only
Hexotic			X		For HexoticPLUGIN mesh only
sphinx		X			
expat	X		X		For YACS only
libBatch		X		X	Required only if used at compilation step
ParaView	X		X		For PARAVIS module only
Homard			X		For HOMARD module only
cgns		X		X	For SMESH only Required only if used at compilation step
freetype		X		X	Required only if used when building OCCT
ftgl		X		X	Required only if used when building OCCT
freeimage		X		X	Required only if used when building OCCT
gl2ps		X		X	Required only if used when building OCCT
Intel TBB		X		X	Required only if used when building OCCT

*) Included to the Distene MeshGems suite v1.0.1 package



HOW TO INSTALL AND BUILD SALOME

- **Linux**

Please follow README file from Installation Wizard for processing correctly installation of SALOME and all prerequisites.

If you would like to compile SALOME from scratch, please use `build.csh` or `build.sh` script delivered with the Installation Wizard. Call "`build.sh -h`" to see all available options of this script.

- **Windows**

For Windows platform SALOME is distributed in form of ZIP archive. To install SALOME on Windows, unpack the archive and follow instructions listed in the enclosed README file.



SALOME SYSTEM REQUIREMENTS

Minimal Configuration:

- Processor: Pentium IV.
- 512 Mb RAM.
- Hard Drive Space: 1.5 Gb.
- Video card 64mb.
- CD/DVD

Optimal Configuration:

- Processor: Dual Core.
- 2 Gb RAM.
- Hard Drive Space: 5Gb.
- 2Gb Swap.
- Video card 128mb.
- CD/DVD



HOW TO GET THE VERSION AND PRE-REQUISITES

SALOME 6.6.0 pre-compiled binaries for Linux Mandriva 2008 (32bits and 64bits), Mandriva 2010 (32bits and 64bits), Debian 4.0 Etch (32bits and 64bits), Debian 5.0 Lenny 64bits, Debian 6.0 Squeeze 64bits, Red Hat Enterprise 4.0 64bits, Scientific Linux 5.1 64bits, CentOS 5.5 64bits and Windows XP 32bits can be retrieved from the SALOME web site <http://www.salome-platform.org>.

The SALOME Installation procedure for Linux includes SALOME modules sources, and it is possible to build sources from scratch using `build.sh` or `build.sch` script coming with installation procedure.

SALOME Installation procedure for Linux includes a patch for **NETGEN** which is placed inside NETGENPLUGIN modules sources. This patch is used for all platforms to fix several bugs of NETGEN. During the compilation on NETGEN from sources by the SALOME Installation Wizard, the patch is applied automatically to the standard NETGEN distribution. You can download NETGEN 4.9.13 from its official site using the following link: <http://www.hpfem.jku.at/netgen>.

All other pre-requisites can be obtained either from your Linux distribution (please be sure to use a compatible version) or from the distributors of these pre-requisites (for example, <http://qt.nokia.com> for Qt). Note that for some of pre-requisite products SALOME Installation procedure also includes patches that fix the problems detected by SALOME.



KNOWN PROBLEMS AND LIMITATIONS

- The following modules have not been migrated to Qt series 4 and thus are not included into SALOME 6.6.0 release: FILTER, SUPERV, MULTIPR. These modules are considered obsolete and not supported anymore.
- Application crash might occur on the data publication in the study if both data server and CPP container are running in the standalone mode.
- On some platforms the default font settings used in SALOME might cause bad application look-n-feel. This problem can be solved by changing the font settings with `qtconfig` utility included into the distribution of Qt 4.
- The following limitations refer to BLSURF plug-in:
 - Mesh contains inverted elements, if it is based on a shape, consisting of more than one face (box, cone, torus...) and if the option "Allow Quadrangles (Test)" has been checked before computation.
 - SIGFPE exception is raised after trying to compute a mesh based on a box with "Patch independent" option checked.
 - It has been found out that BLSURF algorithm can't be used as a local algorithm (on sub-meshes) and as a provider of low-level mesh for some 3D algorithms because BLSURF mesher (and, consequently, the plug-in) does not provide information on node parameters on edges (U) and faces (U, V). For example, the following combinations are impossible:
 - global MEFISTO or Quadrangle(mapping) + local BLSURF;
 - BLSURF + Projection 2D from faces meshed by BLSURF;
 - local BLSURF + Extrusion 3D.
- Sometimes regression test bases give unstable results; in this case the testing should be restarted.
- A native VTK can be used only after manual recompilation with the GL2PS component.
- NETGEN 1D-2D and 1D-2D-3D algorithm do not require definition of 2D and 1D algorithms and hypotheses for both mesh and sub-mesh. 2D and 1D algorithms and hypotheses defined with NETGEN 1D-2D or 1D-2D-3D algorithm will be ignored during calculation.
- SALOME supports reading of documents from earlier versions but the documents created in the new version may not open in earlier ones.
- If SALOME modules are not installed in a single folder, SALOME may not work in the CSH shell since the environment variables are too long by default. In this case, it is suggested to use SH or to install all modules in the same folder.
- During the compilation of OCT 6.x by Makefiles on a station with NVIDIA video card you can experience problems because the installation procedure of NVIDIA video driver removes library `libGL.so` included in package `libMesaGL` from directory `/usr/X11R6/lib` and places this library `libGL.so` in directory `/usr/lib`. However, `libtool` expects to find the library in directory `/usr/X11R6/lib`, which causes compilation failure (See `/usr/X11R6/lib/libGLU.la`). We suggest making symbolic links in that case using the following commands:


```
ln -s /usr/lib/libGL.so /usr/X11R6/lib/libGL.so
ln -s /usr/lib/libGL.la /usr/X11R6/lib/libGL.la
```
- VISU module does not support timestamps defined on the same field but on different meshes
- Stream lines presentation cannot be built on some MED fields due to limitations in VTK.
- MEFISTO algorithm sometimes produces different results on different platforms.
- In some cases the number of triangles generated by MEFISTO may be different at each attempt of building the mesh.

- When generating a 2D mesh with “Maximum Area” hypothesis used, MEFISTO algorithm can produce cells with maximum area larger than specified by the hypothesis.
- For the current moment, because of the ParaView application architecture limitations, PARAVIS module has the following known limitations:
 - PARAVIS is a “singleton” module: that means that it can be used within one study only. As soon as the user activates the PARAVIS in a study, this module becomes unavailable in other studies.
 - PARAVIS module works unstably using a remote connection; when SALOME is running on a remote computer, activation of PARAVIS module can sometimes lead to the application hang-up.
 - PARAVIS module compilation can fail on 64bits platforms when building ParaMEDCorba plugin (due to crash of kwProcessXML tool during generation of the plugin’s documentation). In such a case it is necessary to unset the VTK_AUTOLOAD_PATH environment variable and restart compilation, for example:

```
[bash%] unset VTK_AUTOLOAD_PATH
```