SALOME 7.2.0

Major release announcement

May 2013

❖ **GENERAL INFORMATION**

CEA/DEN, EDF R&D and OPEN CASCADE are pleased to announce SALOME version 7.2.0. It is a major release that contains the results of planned major and minor improvements and bug fixes against SALOME version 6.6.0 released in December 2012.

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❖ NEW FEATURES AND IMPROVEMENTS

PREREQUISITES CHANGES

Important: SALOME 7.2.0 provides results of porting to the latest versions of pre-requisite products. Most important changes are related to the migration on ParaView 3.98 / VTK 6.0. The table below provides the full list of pre-requisite products used with SALOME 7.2.0. The table shows the differences of 3rd-party product versions used for SALOME 7.2.0 and 6.6.0; the changes are highlighted in bold.

Product	SALOME 6.6.0	SALOME 7.2.0
Boost	1.49.0	**1.52.0**
Cgns	3.1.3-4	3.1.3-4
CMake	2.8.7	**2.8.10.2**
Docutils	0.8.1	**0.10**
Doxygen	1.8.0	**1.8.3.1**
Expat	2.0.1	2.0.1
Freeimage	3.14.1	**3.15.4**
freetype	2.4.10	**2.4.11**
Ftgl	2.1.3-rc5	2.1.3-rc5
gl2ps	1.3.5	**1.3.8**
Graphviz	2.28.0	**2.30.0**
HDF5	1.8.8	**1.8.10**
Homard	10.5	**10.6**
Intel® Threading Building Blocks	3.0	3.0
Jinja2	2.6	2.6
LAPACK	3.3.0	**3.4.2**
libBatch	1.6.0	**2.0.0**
Libxml2	2.7.8	**2.9.0**[1]
Med	3.0.6	3.0.6
METIS	4.0	4.0
NETGEN	4.9.13	4.9.13[2]
NumPy	1.5.1	**1.6.2**
omniORB	4.1.6	4.1.6
omniORBpy	3.6	3.6
omniNotify	2.1	2.1
Open CASCADE Technology	6.5.4	**6.5.5**[3]
ParaView	3.14	**3.98.1**[4]

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1 Patched for SALOME (bug for 64bit platforms)
2 Patched for SALOME
3 Patched for SALOME
4 Patched for SALOME (numerous patches of build procedure)

Product	SALOME 6.6.0	SALOME 7.2.0
Pygments	1.5	1.5
PyQt	4.9.1	**4.9.6**
Python	2.6.6	**2.7.3**
QScintilla	2.6.1	**2.7**
Qt	4.6.3	**4.8.4**
Qwt	5.2.1	5.2.1
Scotch	5.1.11	5.1.11
Setuptools	0.6c11	0.6c11
SIP	4.13.2	**4.14.2**
Sphinx	1.1.3	1.1.3
SWIG	1.3.40	**2.0.8**
Tcl	8.5.8	**8.6.0**
Tk	8.5.8	**8.6.0**
TclX	8.4.0	**8.4.1**
VTK[5]	5.9.0	**6.0**
Xdata	0.9.6	**0.9.9**
Distene Blsurf[6]	MeshGems suite v1.0.1	MeshGems suite v1.0.1
Distene TetMesh-GHS3D[7]	MeshGems suite v1.0.1	MeshGems suite v1.0.1
Distene Hexotic[8]	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 and QScintilla (by Riverbank Computing Ltd) used by SALOME are distributed under the terms of GPL license, for more details please refer to:
  - http://www.riverbankcomputing.com/software/pyqt/license
  - http://www.riverbankcomputing.com/software/qscintilla/license

  If you plan using SALOME for commercial usage please consider obtaining a commercial license for PyQt and/or QScintilla.

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5 Version, included in ParaView distribution
6 Commercial product, requires license for using in runtime
7 Commercial product, requires license for using in runtime
8 Commercial product, requires license for using in runtime

MAJOR CHANGES

Major changes introduces by SALOME version 7.2.0 are briefly described in this chapter.

Removal of med v2.1 support

MED files of version 2.1 and older are no more supported by SALOME. These files cannot be imported to SALOME. If necessary, old MED files can be converted to the newer version of format by means of `medimport` utility supplied with med 3.0 library.

Removal of MEDMEM API

MEDMEM and all corresponding API has been removed from SALOME MED module (including CORBA interfaces). Obsolete MEDMEM API has been replaced by MEDCoupling/MEDLoader.

Porting to VTK 6 / ParaView 3.98

SALOME GUI, including ParaVis module, has been migrated to the latest versions of ParaView and VTK.

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MAIN IMPROVEMENTS

Redesigned Salome Notebook

Salome notebook has become a dockable window; by default it shares left dock area of the main window with Object Browser. Notebook window can be closed/opened from the main menu via **View ➔ Windows ➔ Notebook**.

Here it is possible to add new variables, remove and rename existing variables and change their values.

If you have modified some variables, which are already used in the current study, you should click **Update Study** button to apply your changes to the study.
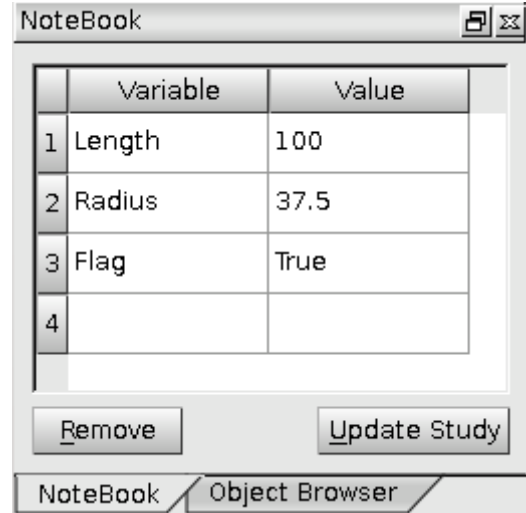
Thickness Reduction on PipeTShape

It has become possible to create thickness reductions at the open ends of a *PipeTShape* in the tab **Thickness reduction** of the dialog box.

For each end it is possible to define:
- The *Radius* of the reduced pipe.
- The *Width* of the reduced pipe.
- The *length of transition* between the pipe and the reduced pipe.
- The *length of the reduced pipe*.

The corresponding parameters have also been added in TUI MakePipeTShape* functions.

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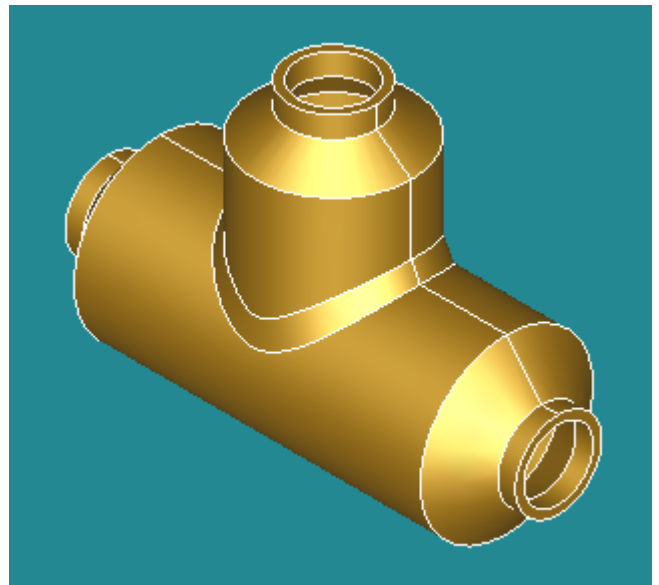
BiQuadratic and TriQuadratic elements

Two new quadratic element types have been introduced in Mesh modules:

- **QUAD9** (a *bi-quadratic* element), which is a quadrilateral quadratic element having an additional node at the element center, and

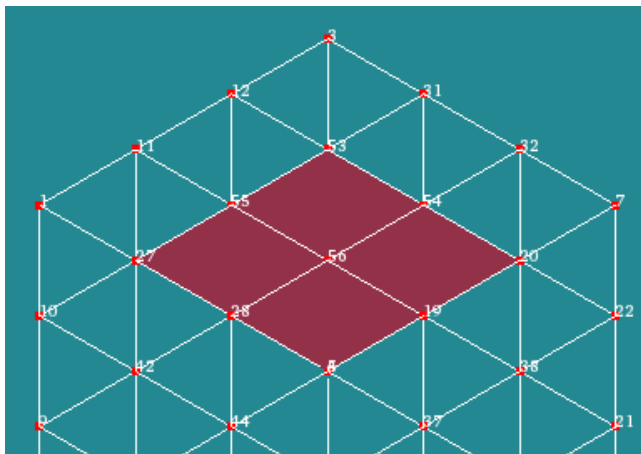- **HEXA27** (a *tri-quadratic* element), which is a quadratic hexahedral element having 7 additional nodes: one at the element center and one at the center of each of six faces.

A Bi-Quadratic element on a box face
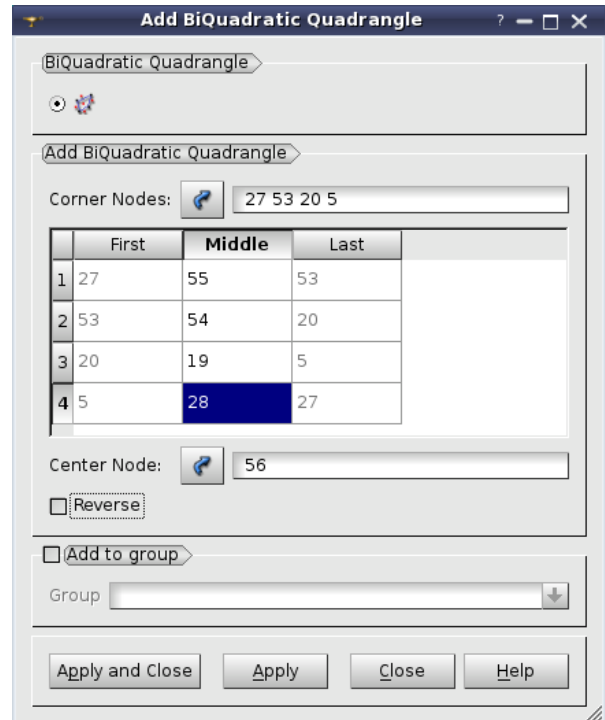
The dialogs for definition of QUAD9 and HEXA27 elements have additional fields **Center node** and **Mid-face Nodes** (for HEXA27 elements only).

Easy-to-use 3D Extrusion
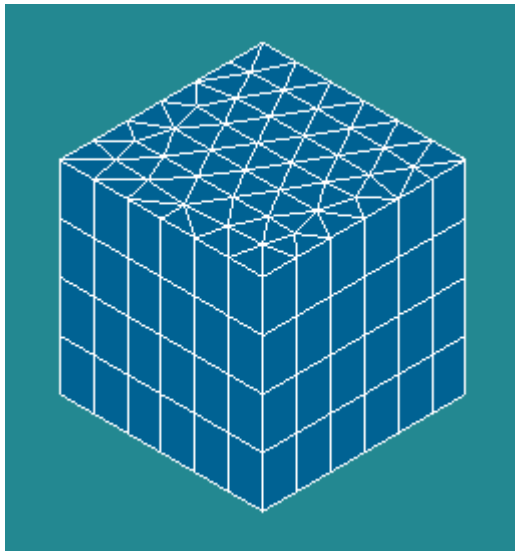
The use of 3D Extrusion algorithm has been facilitated. To obtain the shape shown in the picture at the left it is enough to do the following:

- Define 3D Extrusion algorithm for the main shape (box)

- Define 1D algorithm and hypothesis to divide vertical edges of the main shape

- Create a sub-mesh on the base face of the main shape (in our case it is Automatic triangulation)

Meshing of side faces is now done automatically.

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Minimum Distance

It is now possible to find and publish in the study all possible solutions for Minimum distance.
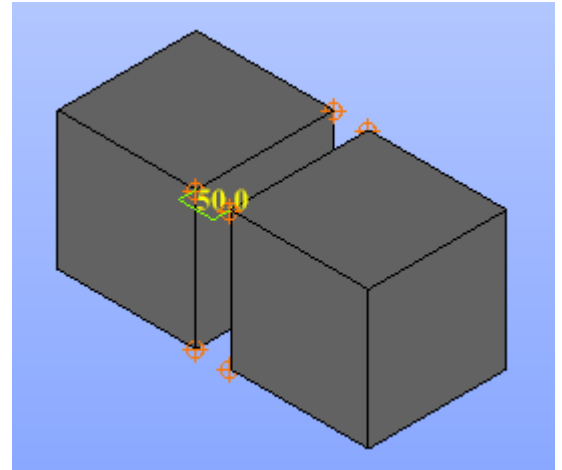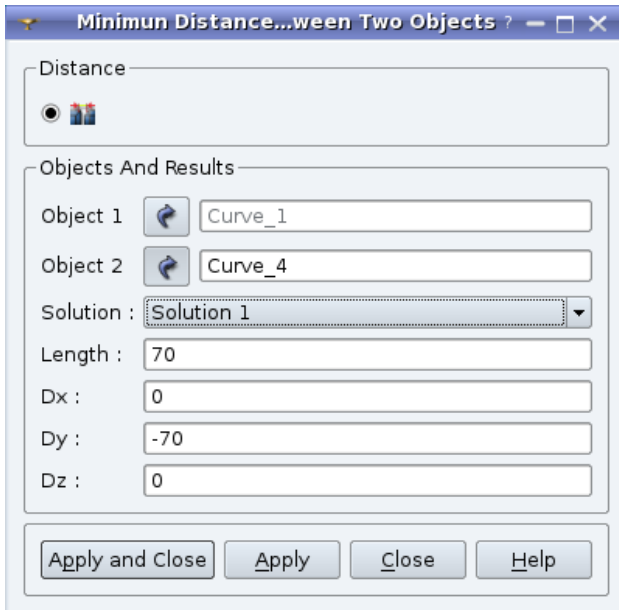
In the image the selected solution between two boxes is marked in green and the distance is indicated in yellow.

The field **Solution** allows selecting the required one from the list of possible solutions.

On clicking "Apply" or "Apply and Close", the following question is shown:
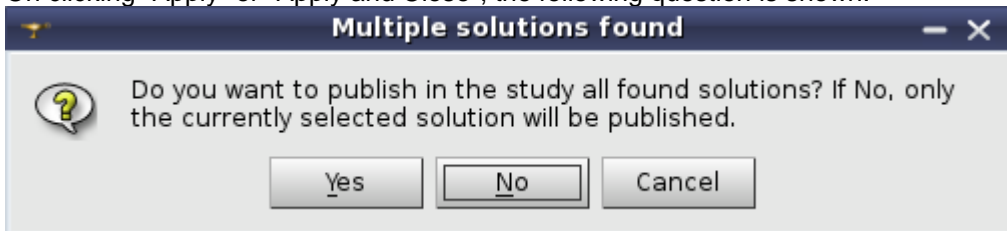
It is possible to publish only one solution (selected in the combo box) or all solutions.

In TUI `geompy.ClosestPoints` function returns the number of found solutions and the coordinates of all points, where the minimal distance is reached: `[nbSolutions, (x11, y11, z11, x21, y21, z21)]`.

Tangency constraints at the ends of interpolation curve

Tangency constraints at the ends of curve have been implemented for curve construction by Interpolation.

In the image, the red curve is constructed only by points and the orange curve is constructed with the impact of First and Last vectors (in white).
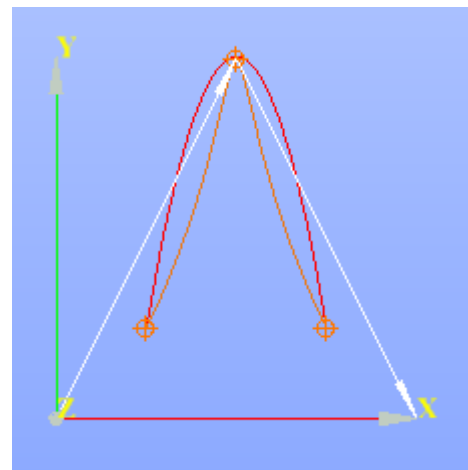
Projection of Rectangular 2D mesh on Conical/Cylindrical/Spherical surface

It has become possible to use mesh on a rectangular face for meshing conical, cylindrical and spherical faces using **Projection 2D** algorithm.

See the examples:
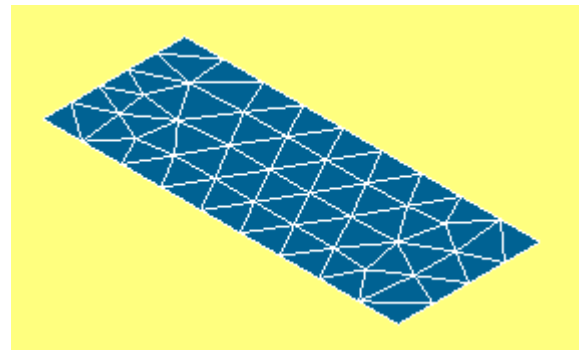
The source rectangular face.
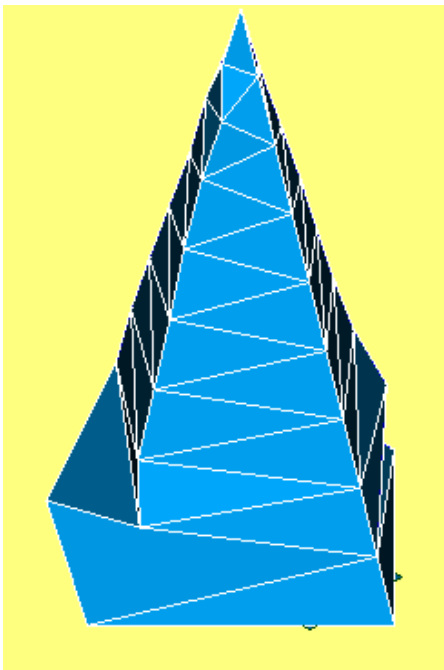
Projection on a cone
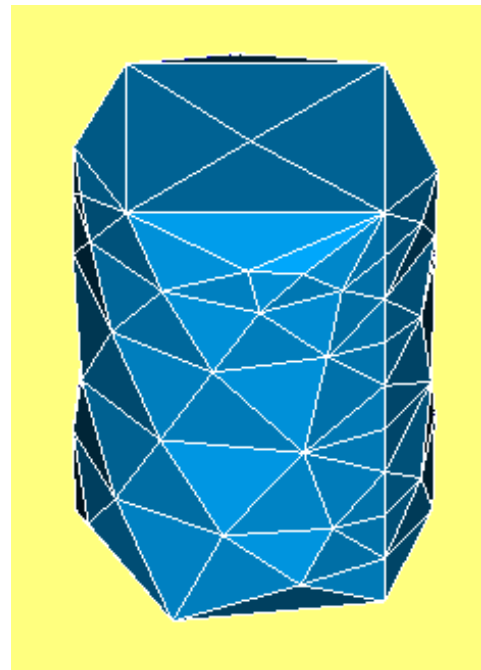
Projection on a cylinder

Improvements in Materials

The same material now looks differently for front and back faces (see the example of a cylinder face), as the property of the material for front and back faces can be specified separately.

Additionally:

- **Materials Library** is now available from the **Tools** menu for editing the existing and defining new materials.

- The predefined material can be assigned to a chosen object from the context menu.
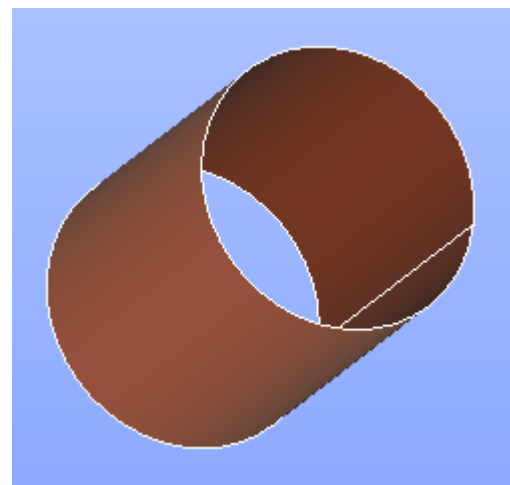
Automatic Bounding Box
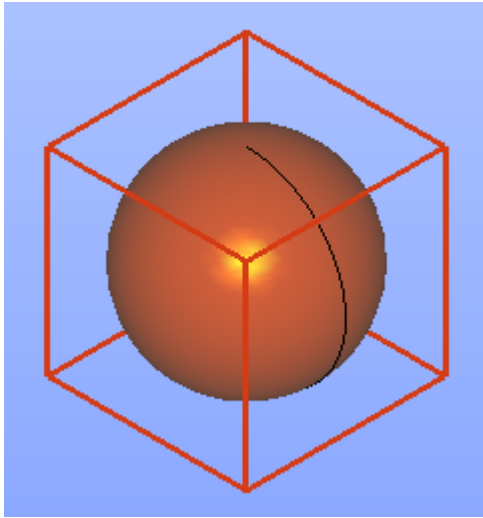
It has become possible to create the bounding box of any object via **Measures ➔ Dimensions ➔ Bounding Box** in Geometry module.

Memory Leaks in Mesh module

The memory leaks related to SMESH data structure and to the use of CORBA have been eliminated from Mesh module.

If the user deletes all meshes and closes a study, the memory occupied by SALOME is not returned to the system because of memory fragmentation. Though the allocated memory is not used, it cannot be returned to the system because pages of memory, even almost fully not used, contain not freed blocks.

The effect of the modifications made can be seen when a mesh is repeatedly computed and cleared (without being shown). In this case, the occupied memory is stable and does not increase constantly. Showing The mesh results in constant growth of occupied memory because of high fragmentation, while the memory checking tool (e.g. Valgrind) shows no leaks.

The following code demonstrates the memory fragmentation effect. A program built from this code does not return memory to the system until termination and constantly occupies about 145 Mb.

```cpp
#include <iostream>

int main (int argc, char ** argv)
{
  int i;
  const int nb = 100000;
  while (true)
  {
    std::cout << "ALLOC" << std::endl;
    char * pointers[nb];
    for(i = 0; i < nb; i++)
      pointers[i] = new char[1024];
    sleep(7);

    // free all but the last chunk
    for(i = 0; i < nb-1; i++)
      delete [] pointers[i];
    std::cout << "FREE" << std::endl;
    sleep(7);

    delete [] pointers[nb-1];
  }

  return 0;
}
```

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Automatic publishing of GEOM objects

The possibility to automatically publish geometry objects created using TUI (Python API) functions in the study has been introduced in GEOM module.

By default, all GEOM Python API functions do not publish resulting geometrical objects. This can be done in the Python script means of `addToStudy()` or `addToStudyInFather()` functions. However, it is possible to publish result data in the study automatically. For this, almost each GEOM Python API function has n additional `theName` parameter (which is None by default). As soon as non-empty string value is passed to this parameter, the result object is published in the study automatically.

For example, consider the following Python script:

```
import salome
from salome.geom import geomBuilder
geompy = geomBuilder.New(salome.myStudy)
box = geompy.MakeBoxDXDYDZ(100, 100, 100) # box is not published in the study
geompy.addToStudy(box, "box")              # explicit publishing
```

Last two lines can be replaced by one-line instruction:

```
box = geompy.MakeBoxDXDYDZ(100, 100, 100, theName="box")
```

… or simply

```
box = geompy.MakeBoxDXDYDZ(100, 100, 100, "box")
```

Automatic publication of all results can be also enabled/disabled by means of the function `addToStudyAuto(maxNbSubShapes=-1)`. The automatic publishing is managed by the numeric parameter passed to this function:
- if `maxNbSubShapes` = 0, automatic publishing is disabled.
- if `maxNbSubShapes` = -1 (default), automatic publishing is enabled and maximum number of sub-shapes allowed for publishing is unlimited; any negative value passed as parameter has the same effect.
- if `maxNbSubShapes` is any positive value, automatic publishing is enabled and maximum number of sub-shapes allowed for publishing is set to specified value.

When automatic publishing is enabled, it is not obligatory to pass `theName` parameter to the functions creating objects; instead default names will be used. However, the behavior can be always changed by passing value to the `theName` parameter; in such a case it will be used instead default one.

For more details, please refer to the documentation of SALOME Geometry module.

OTHER IMPROVEMENTS

- A warning message is now shown in Mesh and Post-Pro modules when a .med file with descending connectivity is read.
- Logarithmic scaling has been introduced in the scalar bar for Mesh quality controls.
- It has become possible to set display mode and color before showing the shape in TUI scripts and thus to directly display a shape in a specific mode or in a specific color.
- The Python examples for GEOM and SMESH Python interface documentation are now generated from real scripts. These scripts are included to the documentation in verbatim form plus the user can save the script to the file directly from the documentation page by pressing "Download this script" link.
- MEDPARTITIONER tool has been improved.
- Display of subshapes and IDs while editing a group has been improved in Geometry module. The possibility to define a specific color for subshapes via preferences has been added.

o Extrusion of a mesh now creates a group of elements not only on the sides of the extrusion (parallel to the direction of extrusion), but also on the face opposite to the base of extrusion.

o It has become possible to move the whole graph into ForEachLoop or OptimizerLoop node using the new submenu "Put Graph Content in Node" in the "Proc menu" context menu.

o The ergonomics of multi-translation and multi-rotation dialogs has been enhanced.

o The results of "Check free boundaries" measurement can be published in the object browser by clicking "Apply" button.

o "Element Info" tab of "Mesh Information" dialog box now provides information about groups, to which the node or element belongs.

o Developer documentation of MED module has been significantly improved for MEDCoupling and MEDLoader packages.

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❖ BUG CORRECTIONS

GUI MODULE

21686	*Summary:* EDF 2303 GUI: Move the notebook window below the OB Salome notebook has become a dockable window shared with the Object Browser. It can be closed/opened from the main menu via "View > Windows > Notebook".
22006	*Summary:* [CEA 727] Restore GUI state makes SALOME crash A crash at the attempt to restore GUI state caused by improper selection management has been fixed.
22092	*Summary:* [CEA 761] 'segmentation violation' when closing salome Fixed SIGSEGV caused by implementation of issue 0021686.
22105	*Summary:* EDF 2561 GUI : Performance regression in the expand all operation Fixed bug in Object browser that leaded to bad performance of "expand all" operation.

GEOM MODULE

21390	*Summary:* EDF 1627 GEOM: Automatic addToStudy The possibility to automatically publish geometry objects created using TUI (Python API) functions in the study has been introduced in GEOM module.
21391	*Summary:* EDF 1630 GEOM: Specific color when editing a group The shapes already present in the edited group are now displayed in a specific color to make difference with the shapes that can be, but are not yet added in the group. Their IDs are also displayed in a specific color in the dialog box, which allows quickly making the difference between the IDs of newly added elements and others.
21394	*Summary:* EDF 1637 GEOM: Function `MinDistance` with creation of vertices "Minimum Distance" operation between two geometrical objects now returns all possible solutions. In TUI new function `geompy.ClosestPoints(Shape1, Shape2)` allows finding all closest points between two shapes in contrast to the existing `geompy.MinDistance` function, which returns only one solution in any case.
21395	*Summary:* EDF 1702 GEOM: Display of GEOM object in 3D viewer while editing a group When a group on shape is edited, the main shape appears in the 3D viewer in the same mode as it is displayed in the viewer. If the main shape is not displayed in the viewer, then the default mode defined in the preferences is used.

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21716	*Summary:* EDF 1679 : Create the bounding box in the GUI It has become possible to create the bounding box of an object via "Measures" > "Dimensions" > "Bounding Box". New python method `MakeBoundingBox(aShape)` returns a `GEOM_Object` with box shape.
21717	*Summary:* EDF 1718: Add thickness reduction to the Pipe T-Shape The PipeTShape functionality of SALOME Geometry module has been extended with additional parameters for adding a thickness reduction at the three extremities (left, right and incident). Each thickness reduction consists of the transition and the thin part. The latter can be omitted if its length is set to zero. The corresponding GUI dialog has been redesigned and split into three tabs: "Main parameters", "Position" and the new tab "Thickness reduction". TUI Python functions `MakePipeTShape*` have been enriched with 12 new parameters, all having zero value by default (i.e. the thickness reductions are switched-off by default). During Python dump operation, only non-zero parameters are dumped.
21718	*Summary:* EDF 1874: Ergonomic of multi-translation / multi-rotation The ergonomics of multi-translation and multi-rotation dialogs has been enhanced. - Some labels have been renamed for better understandability; - Default values are now computed with respect to the dimensions of the transformed shape to improve the preview. - The parameter "Angular Step" is now available for both 1D and 2D multi-rotation. In `geompy` python interface the outdated methods `MultiRotate1D` `MultiRotate2D` `MakeMultiRotation1D` `MakeMultiRotation2D` have been replaced with `MultiRotate1DNbTimes` `MultiRotate1DByStep` `MultiRotate2DNbTimes` `MultiRotate2DByStep` `MakeMultiRotation1DNbTimes` `MakeMultiRotation1DByStep` `MakeMultiRotation2DNbTimes` `MakeMultiRotation2DByStep`
21719	*Summary:* EDF 1654 : Materials The following improvements have been introduced in the Materials definition interface: - Material properties can be defined separately for front and back faces. - Materials library dialog is available to the Tools menu. - Predefined materials can be chosen from the context menu.

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21830	*Summary:* [CEA 652] TUI `createAndDisplayGO` It has become possible to set display mode and color before showing the shape in TUI scripts and thus to directly display a shape in a specific mode or in a specific color. Previously a shape was first displayed in the default mode and color and only then it was possible to change them.
21835	*Summary:* EDF 2070 GEOM : Problem with detecting Self-intersections Fixed two problems with "Detect Self-intersections" on a certain shape, imported from an IGES file: • SIGSEGV on model, imported ignoring units; • A problem of b-spline curves with duplicated poles.
21836	*Summary:* EDF 2204 GEOM : Regression in partition operation The fix for this bug has been made in issue OCC23676. Fixed a problem with Partition between a cylinder and a planar face, touching it by one edge.
21852	*Summary:* EDF 1680 GEOM: Publish "Check free boundaries" results in the Object Browser It has become possible to publish the results of "Check free boundaries" function in the object browser from the corresponding GUI dialog.

21865	*Summary:* [CEA 671] Adding tangency in Spline creation
	
	Tangency constraints at the ends of Interpolation curve have been implemented.
	Two parameters, the First tangent vector and the Last tangent vector, have been added in Curve Construction dialog.
	
	In python this functionality is available via method
	
	`MakeInterpolWithTangents(self, thePoints,`
	`                         theFirstVec, theLastVec)`

21932	*Summary:* [CEA 693] MakeCut function provides an incorrect result between two multi layer spheres The fix for this bug has been made in issue OCC23711. Fixed a problem with Boolean operations on holed solids with solids, filling these holes.
21967	*Summary:* [CEA 707] Partition of a cone with a conic trunk surface The fix for this bug has been made in issues OCC23703, OCC23704. Fixed a problem with Partition of a truncated cone with a conic face.
22009	*Summary:* [CEA 730] `MakePartition` of a solid by an empty compound returns an error The initial shape, which is not modified by a partition operation, is now returned without producing an error.

22012	*Summary:* [CEA 737] An object modified by «Translate» or «Rotate» in the python interpreter is not updated in 3D view An object modified by «Translate» or «Rotate» in the python interpreter is now correctly updated in the 3D view.
22013	*Summary:* [CEA 738] Documentation on `TranslateDXDYDZ` and `Rotate` is wrong about returned object The `geompy` documentation on `TranslateVectorDistance()` and `Rotate()` functions has been corrected.
22051	*Summary:* [CEA 744] SALOME crashes when calliing "BasicProperties" on a sphere degenerated edge Fixed SIGSEGV when building preview on degenerated edge in "Basic Properties" dialog box.
22061	*Summary:* [CEA 747] Glue Edges on ellipsoid and a hole box: operation aborted The fix for this bug has been made in issue OCC23733. Parameter adjustment has been implemented in methods finding extreme points on ellipses to keep the parameter inside the parametric range of the curve.
22062	*Summary:* [CEA 746] A partition between a box and ellipsoids makes Salome crash The fix for this bug has been made in issue OCC23732. Fixed a problem in Partition operation with ellipsoidal arguments.
22095	*Summary:* EDF 2358 GEOM: Add "Create group" in the contextual menu of OCC viewer "Create Group" item has been added in the context menu of OCC viewer. It is accessible if a shape is selected in the viewer.
22126	*Summary:* [CEA 768] Boolean operations on groups: it is not possible to graphically select sub-shapes The following operations with groups have become possible : • Creation of a group of a certain type: vertex, edge, face or solid; • Limitation to add to the group only the objects of the group type; • Create, intersect and cut operations executed only with the chosen group.
22152	*Summary:* EDF GEOM [OCCT 6.5.5 beta]: Regression in a user script due to modifications in the Cut operation The fix for this bug has been made in issue OCC23823. Boolean operation algorithms have been modified to avoid getting redundant vertices in the result of Section.

22164	*Summary:* EDF 2600: A common operation fails The fix for this bug has been made in issue OCC23826. Boolean operation algorithms have been modified to provide correct Common between two solids.
22175	*Summary:* EDF 2607 GEOM: Issue with Remove Extra Edges Remove Extra Edges operation has been improved to work correctly on periodical surfaces.
22177	*Summary:* [CEA 799] RemoveExtraEdges produces non valid faces Remove Extra Edges operation has been improved to work correctly on periodical surfaces.
22181	*Summary:* EDF GEOM: Issue with `MakeRotation` A regression in rotation operations, which concerns the orientation of the edge used as a rotation axis has been fixed.
22186	*Summary:* [CEA 803] `AreCoordsInside` gives a wrong result The check for coordinates has been corrected.
22187	*Summary:* EDF GEOM: Regression with MakePipe The fix for this bug has been made in issue OCC23903. Improved pipe algorithm to glue common faces.

SMESH MODULE

20483	*Summary:* EDF 1117 SMESH,VISU: Mesh with descending connectivity is badly read by SMESH and VISU A warning message is now shown when a .med file with descending connectivity is read in Mesh module.
21397	*Summary:* EDF SMESH: a quadrangle face mesh cannot be projected to a cylinder "Projection 2D" meshing algorithm has been improved to project from the rectangular face to the cylindrical, spherical or conical face.
21556	*Summary:* EDF 2222 SMESH: 3D mesh after projection impossible Wrong position of nodes on edges generated by Projection 2D meshing algorithm has been fixed.

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21840	*Summary:* [CEA 655] Scalar bar: using a logarithmic scale Logarithmic scaling has been introduced in scalar bar for Mesh quality controls. It can be activated by "Logarithmic" checkbox in "Scalar Bar Properties" dialog. This checkbox turns on/off logarithmic scaling of the scalar bar according to the chosen range. If the range contains zero or negative values, the logarithmic scale is disabled. The corresponding Boolean parameter `isLogarithmic` has been added in IDL method `SMESH::NumericalFunctor::GetHistogram()`.
21858	*Summary:* EDF 2171 SMESH : Add an information about the groups, to which an element or node belongs "Element Info" tab of "Mesh Information" dialog box now provides information about groups, to which the node or element belongs. The information on groups can be shown in a short (default) or a detailed mode. The mode can be switched by the "Show details on groups in element information tab" preferences option.
21859	*Summary:* EDF 2191 SMESH : Add conversion from QUAD8 to QUAD9 and from HEXA20 to HEXA27 Bi-quadratic quadrilateral (QUAD9) and tri-quadratic hexahedron (HEXA27) element types have been introduced. "Entity type" criterion has been added to the set of filters for faces to differentiate between QUAD4, QUAD8 and QUAD9 and for volumes to differentiate between HEXA8, HEXA 20 and HEXA27.
21860	*Summary:* EDF 2196 SMESH : Create all the groups in an extrusion operation Extrusion of a mesh now creates a group of elements not only on the sides of the extrusion (parallel to the direction of extrusion), but also on the face opposite to the base of extrusion.
21893	*Summary:* EDF 2133 SMESH : Improvement of 3D extrusion algorithm The usage of "3D Extrusion" algorithm has been facilitated. The 2D hypothesis for the main mesh is always Quadrangle Mapping and side faces of prisms are automatically meshed with quadrilaterals. It is enough to assign the Global 3D Extrusion hypothesis and a 1D hypothesis for the sides of the main shape and to define a custom mesh on the base face.
21948	*Summary:* EDF SMESH : Memory is not freed when deleting a mesh Memory leaks related to SMESH data structure and to the use of CORBA have been eliminated from Mesh module.
21951	*Summary:* EDF 2355 SMESH : Removed "Set Filter" button in the standalone part of the group dialog "Set Filters" button in "Create Group" dialog box has been disabled by default. It can be enabled or disabled by switching on/off "Enable manual edition" checkbox.

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22010	*Summary:* [CEA 731] `CreateDimGroup` creates some faces not belonging to volume skin "Create group of underlying entities" operation has been corrected to avoid adding elements not lying on the elements of input groups.
22082	*Summary:* EDF 1891 SMESH: Performance issues in SMESH filters Performance of mesh highlighting has been optimized.
22089	*Summary:* EDF SMESH 2545: Crash with "Show Edge Direction" in VTK The crash after "Show Edge Direction" in VTK viewer has been fixed. Unnecessary commands have been disabled in the context menu on mesh.
22091	*Summary:* [CEA 760] Regression in test `check_mesh_stability.py` The problem with processing of quadratic elements by "3D Extrusion" meshing algorithm has been fixed.
22123	*Summary:* EDF 2558 SMESH: Group creation crashes with the filter "Free faces" Changed the way the dialog boxes are managed in SMESH module - open dialog windows are not closed when the application window is minimized.
22124	*Summary:* [CEA 763] Projection 1D does not work with Netgen 1D2D The appropriate error message is now shown if the Projection algorithm fails because the source mesh should be computed by an all-dimensional algorithm (such as NETGEN 1D-2D), which, however, would also mesh the sub-shape, to which the Projection algorithm is assigned.
22127	*Summary:* [CEA 765] When using HEXOTIC, mesh computing is OK whereas it does not provide any mesh HEXOTIC plug-in has been corrected to report compute error when necessary.
22130	*Summary:* EDF 2579 SMESH: Issue with Viscous Layer The problem with incorrect 2D Viscous Layers mesh construction if the face has a circular hole has been fixed.
22134	*Summary:* `DriverGMF` : reading quadratic meshes The problems with import and export from/to Distene products of 2D quadratic meshes, which include both linear and quadratic elements, have been fixed by replacing keywords `GmfTrianglesP2`, `GmfQuadrilateralsP2`, etc. with keywords `GmfExtraVerticesAt[ElemType]` where `ElemType` is Edges, Triangles, Quadrilaterals, etc.
22145	*Summary:* [CEA 787] Regression in `test_display.py` `LightApp_Application.cxx` has been modified to avoid critical error at the attempt to restore visual state.

22150	*Summary:* [CEA 789] Projection_1D2D not taken into account by BLSURF Projection 1D-2D algorithm has been improved to compute the mesh on the source sub-shape before the algorithm is launched, which prevents generation of another 1D mesh on the source sub-shape.
22163	*Summary:* SMESH produces study dump with syntax errors The dump procedure has been improved to correctly process scripts containing published objects that depend on non-published ones.
22166	*Summary:* [CEA 792] Regression on 07_Pentahedron.py, 12_3d_extrusion.py and 13_projection_3d.py scripts from CEA test base The regression caused by entering "Convert to bi-quadratic" conversion mode has been fixed.
22182	*Summary:* EDF SMESH: Mesh does not compute after a "Clear Mesh Data" A regression introduced by adjusting `MergeNodes()` for further `Compute()` has been fixed.
22183	*Summary:* EDF SMESH: Error with `ConvertToQuadratic` A regression introduced by adding "Convert to bi-quadratic" conversion mode has been fixed.
22202	*Summary:* EDF SMESH: Regression in a script meshing prisms and hexaedrons The regression entered by the redesign of "3D Extrusion" algorithm has been fixed.
22203	*Summary:* [CEA 814] Regression on test SMESH_AdvancedEditor.py The regression entered by fixing instability of seeping operations (Extrusion, Revolution, etc) has been fixed.
22208	*Summary:* EDF SMESH: Regression / SIGSEGV in an ExtrusionSweepObject operation The regression entered by fixing instability of seeping operations (Extrusion, Revolution, etc) has been fixed.

MED MODULE

21756	*Summary:* [CEA 602] `MEDPartitioner` improvements The following improvements have been introduced in `MEDPARTITIONER` tool: • It has been enabled to work in 2D space; • The option `creates-boundary-faces` has been added. • Sequential and parallel versions of the tool have been enabled.
21856	*Summary:* [CEA 663] Documenting API of `MEDCoupling` and `MEDLoader`. The API of libraries `MEDCoupling` and `MEDLoader` has been documented and the existing documentation has been standardized.

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22080	*Summary:* [CEA 756] Regression: error in libMEDMEM_Swig.MESH function call The command os.makedirs() has been added in the script test_display.py to fix the problem with maillage.ExportMED(med_file).
22151	*Summary:* [CEA 758] doc MED GUI "User guide of the MED Graphical Interface" now can be accessed from the MED documentation.
22168	*Summary:* [CEA 762] MEDLoader documentation An error has been fixed in documentation on MEDLoader API.

PARAVIS MODULE

22125	*Summary:* [CEA 764] Regression in reading file castem.med The problem with the display of families On Cells has been fixed.
22143	*Summary:* [CEA 781] Icon "Save document" stays disabled even when we modify the study The problem with update of the state of "save study" event has been corrected.
22144	*Summary:* [CEA 782] Salome crashes when using "modes" with no Field The python script modes.py has been corrected to avoid crash of the application.
22153	*Summary:* EDF 2264 [PARAVIS]: Progress bar missing' The progress bar indicating the execution of operation has been implemented in Paravis module.
22154	*Summary:* EDF 2508 PARAVIS: Hourglass with "Rescale to Temporal Range" The display of an hourglass during the operations which can take a long time has been restored.
22200	*Summary:* [CEA 813] IW 7.2.0 RC 1: Segmentation fault at SALOME exit if PARAVIS was open Fixed bug of ParaView with closing library with dlclose(). The patch for ParaView has been created.
22204	*Summary:* EDF 2541 PARAVIS : Crash when PARAVIS is loaded after a python component Fixed problem of improper PARAVIS module initialization.

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YACS MODULE

21666	*Summary:* [CEA 562] Put the content of a graph in `ForEachLoop` or `OptimizerLoop` It has become possible to move the whole graph into `ForEachLoop` or `OptimizerLoop` node using the new submenu "Put Graph Content in Node" in the "Proc menu" context menu.

BLSURFPLUGIN MODULE

22011	*Summary:* [CEA 736] Mesh ends with "Global 2D algorithm is missing" A wrong warning on missing algorithm appearing during computation of a mesh with a sub-mesh defined on wires or shells has been eliminated.
22014	*Summary:* [CEA 733] `SetMinSize` has no effect on edges size on arcs "Min Size" and "Max Size" parameters of "BLSURF Parameters" hypothesis are now taken into account by the mesher.
22015	*Summary:* [CEA 734] "Relative value" disappear at hypothesis edition "Relative value" flag of "BLSURF Parameters" hypothesis is now properly stored even if the value of the parameter corresponding to the flag is not changed.
22016	*Summary:* [CEA 740] BLSURF is verbose even with `SetVerbosity (0)` The verbosity level now properly limits the number of messages issued by BLSURF mesher.
22017	*Summary:* [CEA 739] Get an error code returned by `cadsurf_set_param` Validity of parameters passed by BLSURFPlugin to BLSURF mesher is now checked by a returned status value.

NETGENPLUGIN MODULE

21406	*Summary:* EDF 1975 NETGENPLUGIN: Submesh hypothesis not taken into account Fixed regression caused by some of recent changes.
22004	*Summary:* [CEA 725] Remove "Allow quadrangles" from Netgen3D parameters "Allow quadrangles" parameter has been removed from "NETGEN 3D parameters" hypothesis of "Tetrahedron (Netgen)" meshing algorithm.
22005	*Summary:* [CEA 726] Error at the end of NETGEN 3D spheres mesh while tetrahedrons have been generated The problem that caused a failure of NETGEN_3D algorithm when 2D mesh was generated by BLSURF mesher has been fixed.

DOCUMENTATION

21895	*Summary:* EDF 2372 GEOM SMESH : Generate the doc examples from real python scripts The Python examples for GEOM and SMESH Python interface documentation are now generated from real scripts. The examples are located in a new 'doc/salome/examples' directory. It is possible to execute them using make check command. The scripts also can be accessed from the documentation about the corresponding operations.

INSTALLATION PROCEDURE

22050	*Summary:* [CEA 743] With a Salome installed with the Wizard, NSparam returns 2810 even if Salome has not been launched The problem with outdated installation procedure of .omniORB.cfg has been resolved

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❖ PROCESSED AND ANSWERED QUERIES

21131	*Summary:* EDF 1167 NETGENPLUGIN: SIGSEGV when trying to mesh a shape with degenerated edges
21831	*Summary:* EDF 2247 GEOM : Partition fails between a compound and a face
21886	*Summary:* [CEA 677] Netgen 3D error on an overlaping box
22003	*Summary:* [CEA 724] error on a `MakePartition`
22008	*Summary:* [CEA 729] Wrong `MakeCommon` result between a sphere and a box
22055	*Summary:* [CEA 741] One surface edge intersects a surface face
22072	*Summary:* [CEA 751] SIGSEGV by just opening and directly closing PARAVIEW with PARAVIS plugin
22073	*Summary:* [CEA 752] Strange messages on PARAVIEW launch with Salome plugins
22083	*Summary:* Missing nodes on share shapes with Netgen 1D-2D
22085	*Summary:* EDF 2535 SMESH : Issue when displaying a mesh
22090	*Summary:* [CEA 759] Regression on test `MEDLoaderTest3.py`
22132	*Summary:* [CEA 770] Install Wizard for SALOME 64bits seems to be compiled in 32bits
22135	*Summary:* [CEA 775] On BOP version, a `MakeVertexOnCurve` on a degenerated edge makes SALOME crash
22141	*Summary:* [CEA 778] Anomaly in Paravis / Contour filter
22156	*Summary:* EDF 2396 PARAVIS: RunSalomeScript + PARAVIS
22167	*Summary:* [CEA 795] Online documentation: search on "Salome mesh refinement"
22176	*Summary:* EDF 2599 SMESH: Rectangular selection when the trihedron is displayed
22180	*Summary:* EDF GEOM: Issue with MakeBoolean
22185	*Summary:* EDF GEOM: Regression with CreateGroup in order to create a SHELL type group

22188	*Summary:* [CEA 806] regression: `HAMMI_177_MakeSpring_v3.py` problem with a conversion from float to integer
22190	*Summary:* EDF GEOM: Bug with MakeRevolution
22191	*Summary:* [CEA 809] Regression: Stability problem at mesh computation
22193	*Summary:* EDF 2634 [MED]: Segmentation violation with "Visualize/Use in workspace"
22194	*Summary:* EDF SMESH: New groups with `ExtrusionAlongPathObjX`
22196	*Summary:* [CEA 808] On windows version, impossible to rename objects in object browser
22206	*Summary:* EDF 2633 SMESH: BelongToGeom with a compound

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❖ OCCT 6.5.5 BUG CORRECTIONS

SALOME 7.2.0 is based on a patched version of Open CASCADE Technology version 6.5.5 that has been released in March 2013.

This chapter lists bug corrections and improvements made for SALOME and included in Open CASCADE Technology version 6.5.5. The issues included to the patch are described separately.

Note that only the issues connected with SALOME platform are listed below.

BUG FIXES INCLUDED TO OCCT VERSION 6.5.5

OCC23530	*Summary:* Error in Draw command "checkshape": invalid face is not recognized Referenced by 0021777: [CEA 611] RemoveExtraEdges on rounded cylinder
OCC23548	*Summary:* Boolean operation between two faces fails Fix regression detected by test script bugs/L2
OCC23552	*Summary:* Projection algorithm produces wrong results with default tolerance value Referenced by 0021547: EDF 2194 GEOM : The partition btw an ellipsoid and a box gives a bad result
OCC23567	*Summary:* Wrong STEP unit is returned by STEPControl_Reader.FileUnits() function Referenced by 0021511: EDF 2153 GEOM: Choose to take units into account or not when importing a STEP file.
OCC23576	*Summary:* Intersection algorithm produces trimmed circle with illegal parametric range. Referenced by 0021991: [CEA 717] GlueEdges fails because an unexpected point had been created by MakeCut
OCC23603	*Summary:* Boolean operation between two edges fails Fix regression detected by test script bugs/N9
OCC23606	*Summary:* Invalid result of pipe operation Fix regression detected by test script pipe/C0. The issue provides alternative implementation of solution for SALOME issue 0020207: EDF GEOM 960: MakePipe : Generation of redundant doubled faces.
OCC23643	*Summary:* Intersection algorithm produces B-Spline curve in case of coaxial cones Fix regression detected by test script partition_EDF/C9
OCC23644	*Summary:* Intersection between coaxial cones produces exception Fix regression detected by test script partition_EDF/C9

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OCC23676	*Summary:* Intersection between edge and a face produces extra vertex Referenced by 0021836: EDF 2204 GEOM : Regression in partition operation
OCC23699	*Summary:* Exception raised during intersection between two surfaces Referenced by 0021835: EDF 2070 GEOM : Problem with detecting Self-intersections
OCC23703	*Summary:* Projection algorithm produces incomplete 2D-Curve Referenced by 0021967: [CEA 707] Partition of a cone with a conic trunk surface
OCC23704	*Summary:* The program hang-up during intersection of two edges Referenced by 0021967: [CEA 707] Partition of a cone with a conic trunk surface
OCC23705	*Summary:* Isoline in the AIS viewer is not trimmed Referenced by 0021191: EDF 1628 SMESH: Detection of close but not sewed edges
OCC23711	*Summary:* Incorrect result done by fuse operation Referenced by 0021932: [CEA 693] MakeCut function provides an incorrect result between two multi layer spheres
OCC23716	*Summary:* Boolean Fuse between two faces fails Referenced by 0021976: [CEA 712] Regression in the MakeCut method in V6_6_BR version
OCC23732	*Summary:* Exception raised during intersection between two surfaces Referenced by 0022062: [CEA 746] A partition between a box and ellipsoids makes Salome crash
OCC23733	*Summary:* `PCurve` for edge on face creation failure Referenced by 0022061 [CEA 747] Glue Edges on ellipsoid and a hole box: operation aborted
OCC23738	*Summary:* Boolean Fuse between two faces fails Fix regression detected by test script bugs/XPR_01

CONTENTS OF PATCH FOR OCCT VERSION 6.5.5

OCC23775	Summary: Compatibility with Tcl/Tk 8.6 Provide compatibility with tcl/tk 8.6.0 that is required by SALOME 7.2.0.

OCC23823	Summary: The result of section operation contains redundant vertex. Referenced by 0022152: EDF GEOM [OCCT 6.5.5 beta]: Regression in a user script due to modifications in the Cut operation
OCC23826	*Summary:* Boolean Common between two solids fails Referenced by 0022064: EDF 2600: A common operation fails
OCC23903	*Summary:* Invalid result of pipe operation on closed path Referenced by 0022187: EDF GEOM: Regression with MakePipe

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❖ SUPPORTED LINUX DISTRIBUTIONS AND PRE-REQUISITES

SALOME 7.2.0 supports Linux Debian Squeeze 6.0 64bits, Mandriva 2010 32bits/64bits, CentOS 5.5 64bits, CentOS 6.3 64bits, Fedora 17 64bits and Fedora 18 64bits. SALOME 7.2.0 has been tested with the pre-requisites listed in the table below.

SALOME 7.2.0 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	GHS3DPRLPLUGIN	BLSURFPLUGIN	HexoticPLUGIN	HEXABLOCKPLUGIN
gcc*	4.1**	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	2.8.10.2								X								
Python	2.7.3	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X
Qt	*4.8.4*	X		X	X	X	X	X	X	X	X	X	X	X	X	X	X
Sip	4.14.2	X			X												
PyQt	4.9.6	X			X												
Boost	1.52.0	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X
Swig	2.0.8	X	X	X	X	X	X	X		X		X	X	X	X	X	X
OCCT	6.5.5	X		X	X	X	X	X	X	X	X	X	X	X	X	X	X
Qwt	5.2.1	X			X												
QScintilla	2.7								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	1.8.10	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	6.0.0	X		X	X	X	X		X		X	X	X	X	X	X	X
numpy	1.6.2		X														
lapack	3.4.2		X														
graphviz	2.30.0	X	X	X	X	X	X	X				X	X	X	X	X	
Doxygen	1.8.3.1	X	X	X	X	X	X	X				X	X	X	X	X	X
NETGEN	4.9.13											X					
docutils	0.10	X	X	X	X	X	X	X			X	X	X	X	X	X	
metis	4.0							X									
scotch	5.1.11							X									
libxml2	2.9.0	X	X				X	X									
Distene MeshGems	1.0.1												X	X	X		
Hexotic	1.0															X	
sphinx	1.1.3		X	X	X			X					X	X			
expat	2.0.1							X									
libBatch	2.0.0		X														
cgns	3.1.3				X												
ParaView	3.98.1								X								
Homard	10.6									X							

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

Product	Version	RANDOMIZER	SIERPINSKY	PYCALCULATOR	COMPONENT	CALCULATOR	HELLO	LIGHT	PYLIGHT	ATOMIC	ATOMGEN	ATOMSOLV	HXX2SALOME	YACSGEN	JOBMANAGER
gcc*	4.1**	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
autoconf*	2.59**	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
GNU make*	3.80**	X	X	X	X	X	X	X	X	X	X	X	X		X
Python	2.7.3	X	X	X	X	X	X	X	X	X	X	X		X	X
Qt	4.8.4		X		X	X	X	X		X	X	X	X		X
Sip	4.14.2				X						X				
PyQt	4.9.6				X				X		X				
Boost	1.52.0		X		X	X							X		X
Swig	2.0.8		X	X	X										
OCCT	6.5.5		X		X	X	X	X		X		X			
Qwt	5.2.1				X										
OmniORB	4.1.6	X	X	X	X	X	X				X	X			X
OmniORBpy	3.6	X	X	X	X	X	X				X	X			X
Hdf5	1.8.10		X		X	X		X		X					
Med	3.0.6		X	X	X	X									
Vtk	6.0.0		X		X			X	X	X		X			
graphviz	2.30.0	X	X	X	X		X				X				
Doxygen	1.8.3.1	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	8.6.0	Open CASCADE Technology, VTK	Optional
tk	8.6.0	Open CASCADE Technology, VTK	Optional
tclX	8.4.1	Open CASCADE Technology	Optional
jinja	2.6	Sphinx	
pygments	1.5	Sphinx	
setuptools	0.6c11	Sphinx	
freetype	2.4.11	Open CASCADE Technology	
ftgl	2.1.3	Open CASCADE Technology	
freeimage	3.15.4	Open CASCADE Technology	Optional
gl2ps	1.3.8	Open CASCADE Technology	Optional
Intel TBB	3.0	Open CASCADE Technology	Optional
xdata	0.9.9		Can be used to create 3rd- 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 7.2.0 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.

Software Requirements

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		Mandatory for BLSURFPLUGIN only, optional for HEXOTIC plugin
TetMesh-GHS3D*	X		X		For GHS3DPLUGIN and GHS3DPRLPLUGIN 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.

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❖ HOW TO INSTALL AND BUILD SALOME

Please follow README file from Installation Wizard for correct installation of SALOME and all prerequisites on Linux.

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 available options of this script.

❖ SALOME SYSTEM REQUIREMENTS

Minimal Configuration:

- Processor: Pentium IV

- 512 Mb RAM

- Hard Drive Space: 2 Gb

- Video card 64 Mb

Optimal Configuration:

- Processor: Dual Core

- 2 Gb RAM + 2 Gb Swap

- Hard Drive Space: 5 Gb

- Video card 128 Mb

❖ HOW TO GET THE VERSION AND PRE-REQUISITES

SALOME 7.2.0 pre-compiled binaries for Linux Mandriva 2010 32bits and 64bits, Debian 6.0 Squeeze 64bits, CentOS 5.5 64bits, CentOS 6.3 64bits, Fedora 17 64bits and Fedora 18 64bits can be retrieved from the SALOME web site http://www.salome-platform.org or from ftp repository ftp://ftp.opencascade.com.

The SALOME Installation procedure 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 module 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

- SALOME platforms supports two independent build systems – based in GNU auto-tools and on CMake. However, CMake-based build procedure is an experimental one and might not work on some platforms. Also, not all SALOME modules can be built with CMake yet.

- The following modules are not included into SALOME 7.2.0 release: FILTER, SUPERV, MULTIPR, PYCALCULATOR. These modules are considered obsolete and not supported anymore.

- The Post-Pro (VISU) module is considered obsolete. It is replaced by ParaVis module.

- 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;
        - BLSUFR + 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 in general supports reading of documents from earlier versions but the documents created in the new version may not open in earlier ones. However, some studies may work incorrectly in SALOME 7x; mainly it concerns studies with Post-Pro data in which med v2.1 files have been imported. Due to removal of med v2.1 support in SALOME series 7x, there can be problems with opening of such studies in SALOME.

- 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
```

SALOME Platform

- Post-Pro (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:

   o 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.

   o 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.

   o PARAVIS module compilation can fail on 64-bit platforms when building `ParaMEDCobra` plugin (due to crash of `kwProcessXML` tool during generation of the plugin documentation). In such case it is necessary to unset `VTK_AUTOLOAD_PATH` environment variable and restart the compilation, for example:

   `[bash%] unset VTK_AUTOLOAD_PATH`

- There can be problems when installing SALOME with Installation Wizard in "build from sources" mode caused by numerous bugs in ParaView 3.98 build procedure: on some platforms the compilation of ParaView can take more than 24 hours. Moreover, compilation of ParaView on some platforms can even fail.

SALOME Platform