

SALOME version 7.6.0

Public release announcement

July 2015

❖ GENERAL INFORMATION

CEA/DEN, EDF R&D and OPEN CASCADE are pleased to announce [SALOME](#) version [7.6.0](#). It is a public release that contains the results of planned major and minor improvements and bug fixes against SALOME version 7.5.1 released in February 2015.

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❖ NEW FEATURES AND IMPROVEMENTS

PREREQUISITES CHANGES

The table below lists pre-requisite products used with SALOME 7.6.0 the differences of 3rd-party product versions used for SALOME 7.6.0 and 7.5.1 are highlighted in bold.

Product	SALOME 7.5.1	SALOME 7.6.0
Boost	1.52.0	1.52.0
Cgns	3.1.3-4	3.1.3-4
CMake	2.8.10.2	2.8.10.2
Distene MeshGems suite ¹	1.3-8	2.0
Distribute	-	0.6.28
Docutils	0.10	0.10
Doxygen	1.8.3.1	1.8.3.1
Freeimage	3.16.0	3.16.0
Freetype	2.4.11	2.4.11
Gl2ps	1.3.8	1.3.8
Graphviz	2.38.0	2.38.0
HDF5	1.8.10	1.8.10
Homard	11.1	11.1
Intel® Threading Building Blocks	3.0	4.2.4
Jinja2	2.6	2.6
LAPACK	3.5.0	3.5.0
libBatch	2.2.0	2.3.0
Libxml2	2.9.0	2.9.0
Matplotlib	1.3.1	1.4.3
Med	3.0.8p1	3.0.8p1
METIS	4.0	4.0
NETGEN	4.9.13	4.9.13
NumPy	1.8.1	1.8.2
omniORB	4.1.6	4.1.6
omniORBpy	3.6	3.6
omniNotify	2.1	2.1
Open CASCADE Technology	6.8.0	6.9.0²
ParaView	4.2.0	4.3.1³

¹ Commercial product; requires license.

² Patched for SALOME.

³ Patched for SALOME.

Product	SALOME 7.5.1	SALOME 7.6.0
Py-parsing	-	2.0.3
Pygments	1.5	1.5
PyQt	4.9.6	4.9.6
Python	2.7.3	2.7.3
Python-dateutil	-	2.3
Pytz	-	2015.2
QScintilla	2.7	removed
Qt	4.8.4	4.8.4
Qwt	6.1.0	6.1.0
SciPy	0.13.3	0.14.1
Scotch	5.1.11	5.1.11
Setuptools	0.6c11	0.6c11
SimanIO	1.0	1.0
SIP	4.14.2	4.14.2
Six	-	1.8.0
Sphinx	1.1.3	1.1.3
SWIG	2.0.8	2.0.8
Tcl	8.6.0	8.6.0
Tk	8.6.0	8.6.0
TclX	8.4.1	8.4.1
VTK ⁴	6.2	6.2
Wso2-wsf-cpp	2.1.0	2.1.0
Xdata	0.9.9	0.9.9

Note: the table above lists only main pre-requisite products (used with default configuration of SALOME); some optional products are not shown. For additional information about pre-requisite products and SALOME modules dependencies refer to the paragraph “[Supported distributions and pre-requisites](#)” below.

License restrictions

Hereby we explicitly declare that PyQt (by Riverbank Computing Ltd) used by SALOME is distributed under the terms of GNU GPL license; for more details please refer to the PyQt site:
<http://www.riverbankcomputing.com/software/pyqt/license>

If you plan using SALOME for commercial purposes please consider obtaining a commercial license for PyQt from Riverbank Computing Ltd.

⁴ Version included into ParaView distribution.

MAIN IMPROVEMENTS

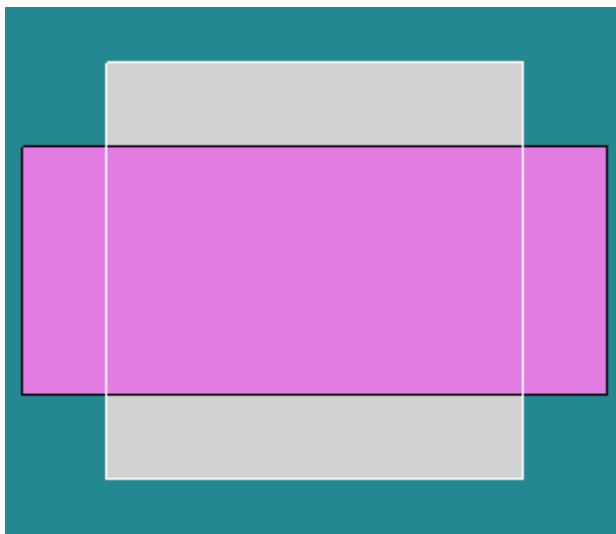
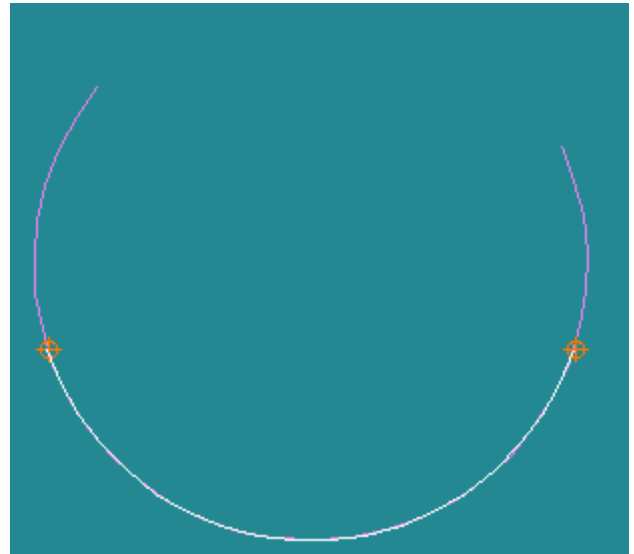
Extend Face or Edge

New **Extension** operation allows resizing edges or faces.

This operation is available via menu **Operations** → **Transformation** → **Extension**

It has become possible to resize an **Edge** by modifying its first and last parameters.

Please, see in the image to the right the original edge in white and extended edge in pink.



It is also possible to resize a **Face** by modifying its minimal and maximal U- and V-Parameters.

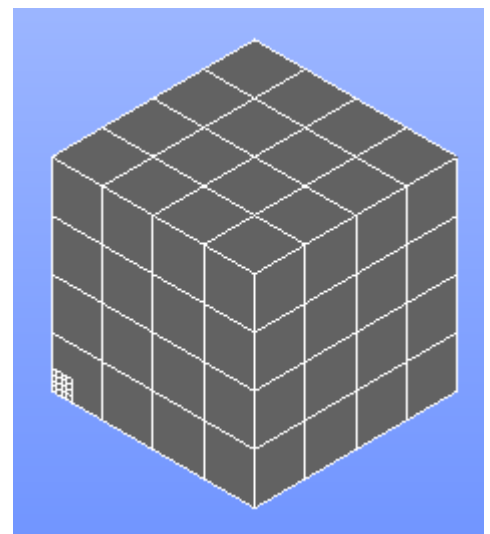
In the image to the left the original face is grey and the result face shrunk along U-Direction and extended along V-Direction.

Remove Small Volumes

Shape processing operation now allows removing small solids or merging them with neighboring ones

The small volumes can be filtered by **Width factor tolerance**, which defines the maximum value of $2V/S$ of a solid, which is considered small, where V is the volume and S is the surface area of the solid or **Volume tolerance**, which defines the maximum volume of a solid, which is considered small.

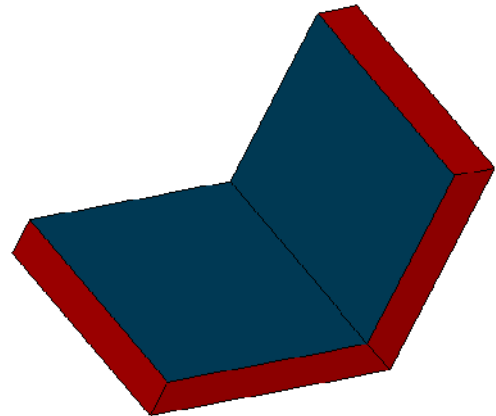
To merge solids checkbox allows removing small solids; else they are merged to adjacent non-small solids or left untouched if they cannot be merged.



Thickness

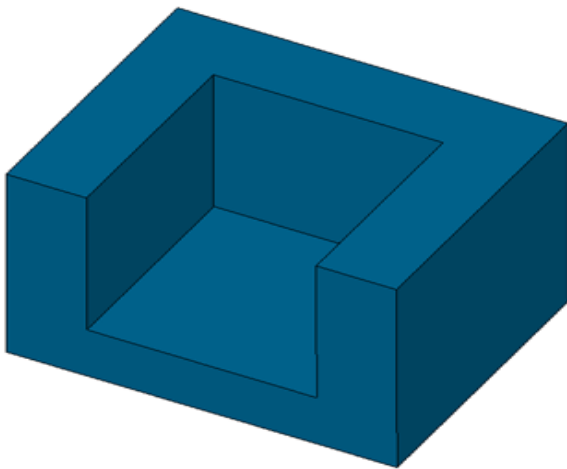
Thickness can be applied to a Face or a Shell to create a Solid. This operation previously available in TUI mode only, has been added to GUI now, in menu **New Entity** → **Generation** → **Thickness**.

To use this operation, it is necessary to define an **Object** (Face or Shell) and the value and the direction of **Thickness**.

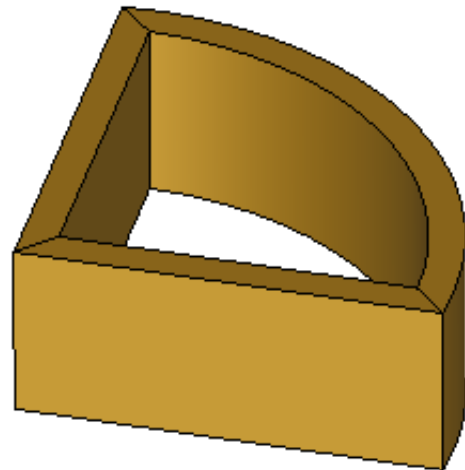


It is also possible to apply the Thickness to a Solid to create a hollowed Solid.

It is necessary to define **Solid Object Faces** to be removed from the result and **Thickness**. **Thicken towards the inside** check box allows changing the thickness direction.



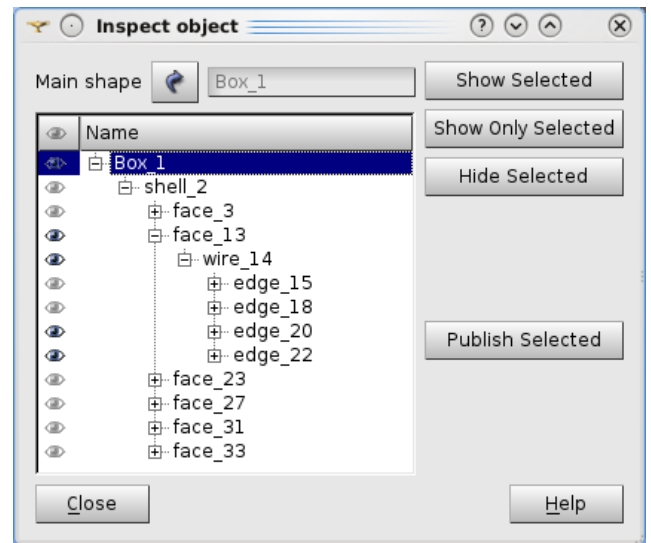
Thickness can be also added to the prism created by extrusion (only when extruding an edge or wire).



Inspect Object

New **Inspect Object** operation available from **Measures** main menu group allows browsing the contents (all sub-objects) of the selected shape.

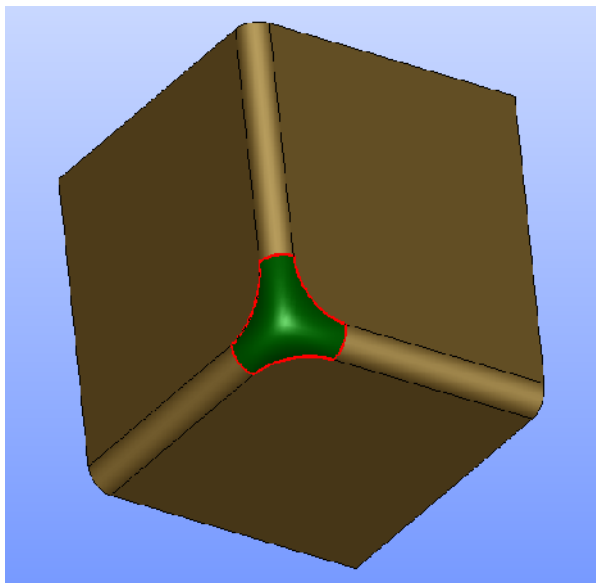
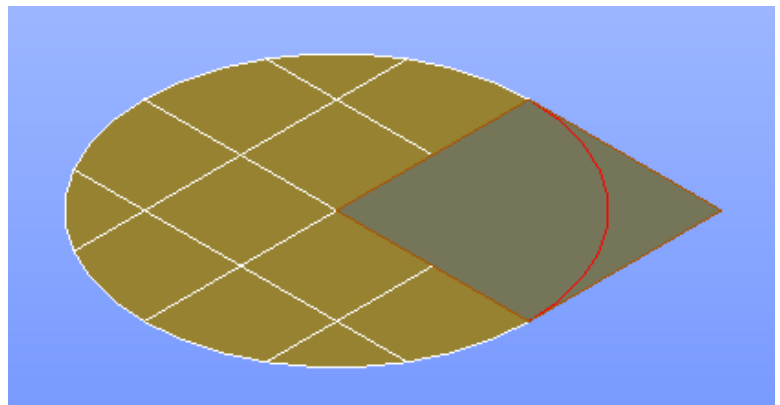
It allows selecting one or several entities in the tree view, changing names, showing/hiding objects in the viewer, publishing them in the study, etc.



New Face Creation Algorithms

It is possible to create a face based on the surface of another face and bound by a wire.

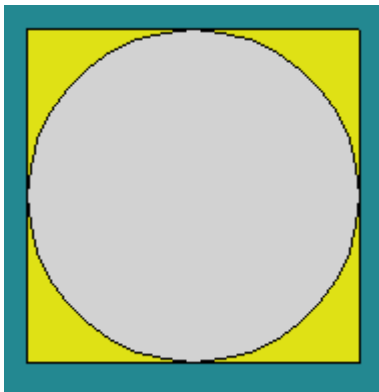
In the image to the right there is a circular face created from the quadrangular one and limited by a circular wire.



It is also possible to create a face from a closed wire and a list of constraints (edges and constraint faces).

The green face to the left results from a red wire with specifying constraint faces (colored light-gold on a picture).

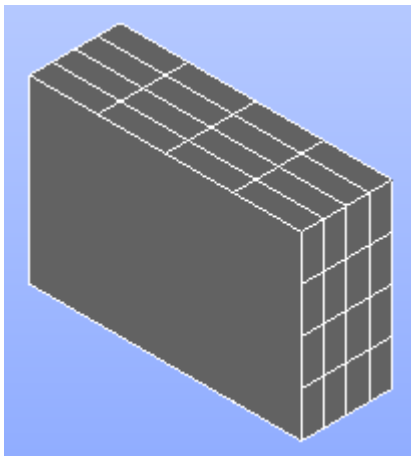
Surface from Face



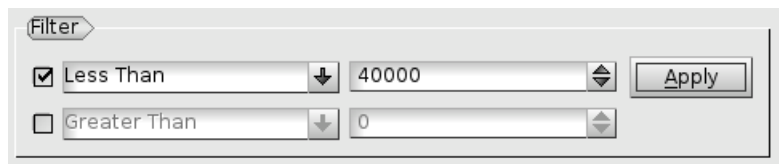
New **Surface from Face** operation takes a face at input and creates a new topological shape by extracting the underlying surface of the source face and limiting it by the **Umin**, **Umax**, **Vmin** and **Vmax** parameters of the source face (in the parametric space).

In the image, the source face is shown in white and the resulting surface in yellow.

Size Filter for group creation

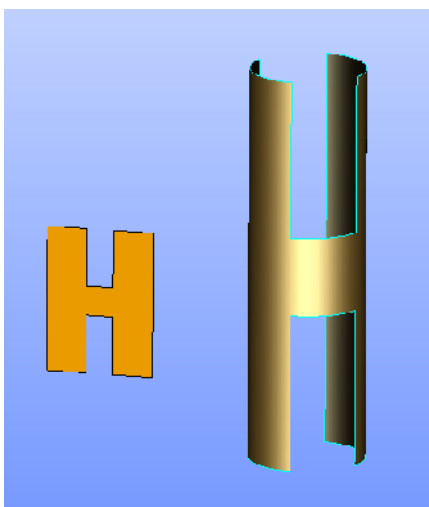


Filter group has been introduced in “Create Group” and “Sub-shapes selection” dialogs to highlight and select entities with parameters (length, area or volume) that are less and/or greater than the given value.



See the filtered faces of the box to the left.

Projection on Cylinder



New operation **Projection on Cylinder** projects a Source planar wire or face on a cylinder defined by its radius. The cylinder’s coordinate system is the same as the global coordinate system. The result is a projection of the source shape wire or a face on the cylinder.

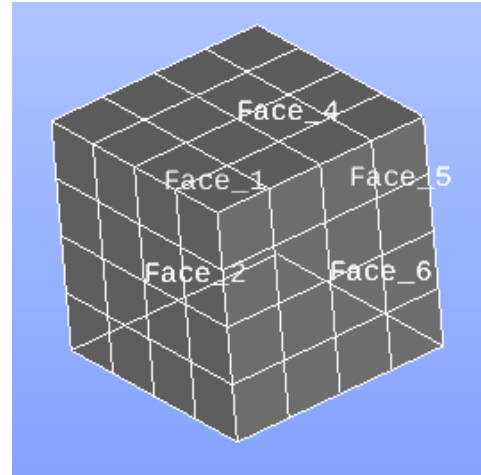
In the image, the source face is shown in orange and its projection on the cylinder in gold.

Names of Objects in the Viewer

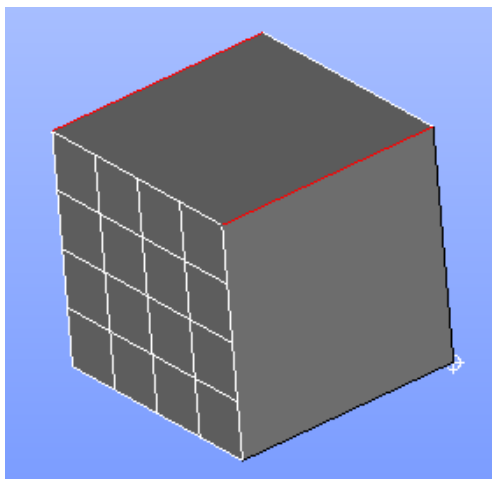
It has become possible to display the name of a geometrical object in OCC and VTK 3D viewers.

This function is available from the context menu of the selected object by choosing **Display Mode** → **Show name**.

The name label is a 2D text that cannot be scaled or rotated. It is bound to the gravity center of the object.



Simultaneous Selection of Different Types of Entities



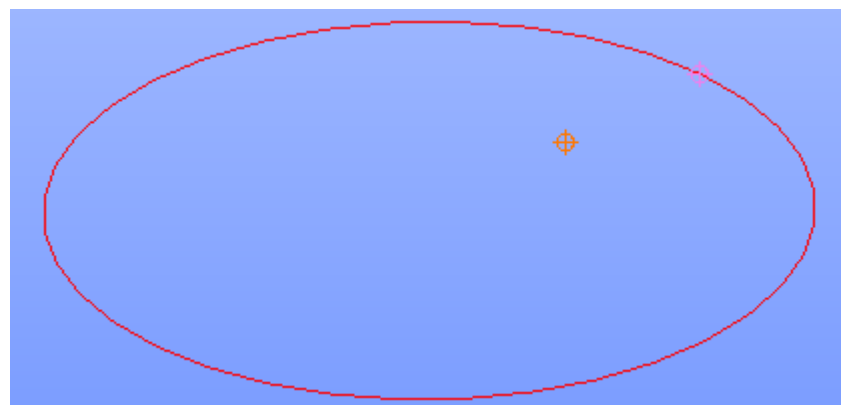
The local selection capabilities of OCC 3D viewer in the Geometry module now allow selection of different types of entities simultaneously.

On the box to the left a face, an edge and a vertex are selected (in white).

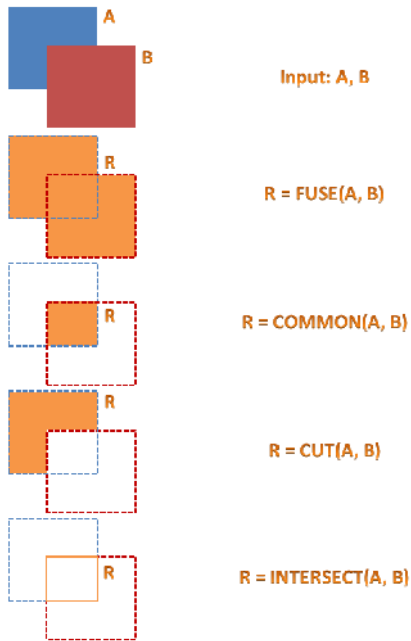
Projection of Vertices on Wires and Edges

Projection dialog now allows projecting vertices to wires and edges in addition to projection on faces.

In the image, a vertex is projected on a circular wire.



Intersection operation



Former Section operation has been renamed into **Intersection**. This operation has been redesigned to allow intersecting different types of shapes and not faces only.

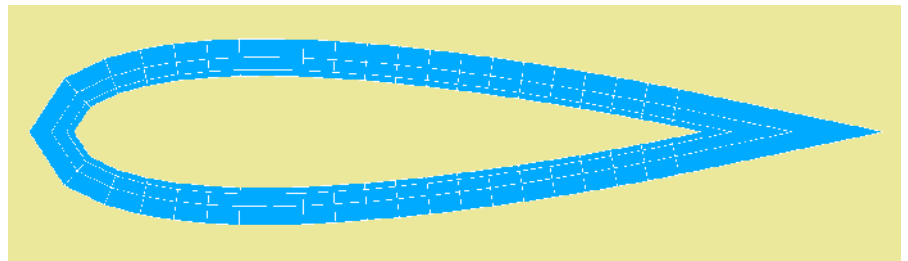
As the result, Intersection operation completes a set of Boolean operations; the image demonstrates the results of different Boolean operations applied to two shapes.

Extrusion by Normal

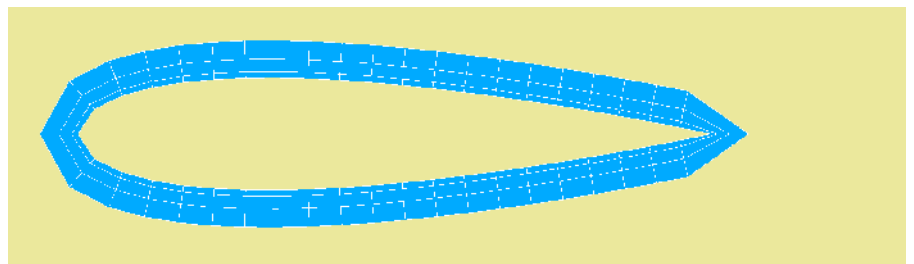
Extrusion by Normal allows extruding every node of selected faces along the *average* of the *normal* vectors to the faces sharing the node (nodes and edges can't be extruded in this mode.)

There are two alternative ways to measure the distance of extrusion:

If **Along average normal** check-box is *activated*, the distance is measured along the average normal.



If it is disabled, every node is extruded along the average normal up to its intersection with the virtual plane obtained by translation of the face sharing the node along its own normal by the **Distance**.



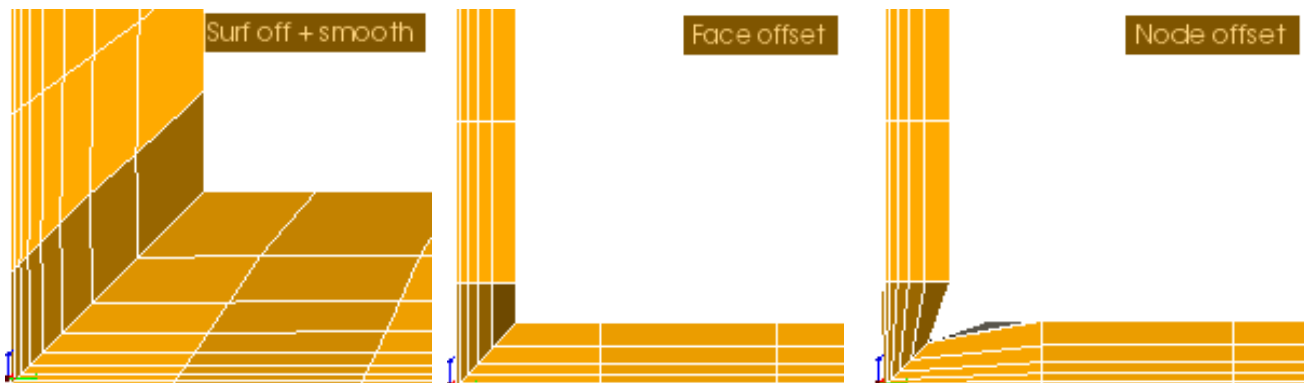
New Extrusion Methods for Viscous Layers

Two new extrusion methods now can be used at viscous layers construction. The extrusion method defines how positions of new nodes are found during prism construction and how creation of distorted and intersecting prisms is prevented.

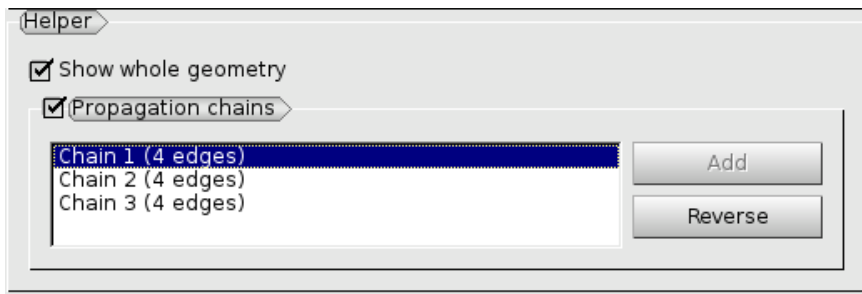
The old method (default) referred to as **Surface offset + smooth** extrudes nodes along the normal to the underlying geometrical surface. The internal surface of element layers can be smoothed to avoid creation of invalid prisms.

The first new method named **Face offset** extrudes nodes along the average normal of the surrounding mesh faces to the intersection with a neighbor mesh face translated along its own normal by the layers thickness. The thickness of layers can be limited to avoid creation of invalid prisms.

The second new method named **Node offset** extrudes nodes along average normal of surrounding mesh faces by the layers thickness. The thickness of layers can be limited to avoid creation of invalid prisms.



Reversed Edges Helper Group

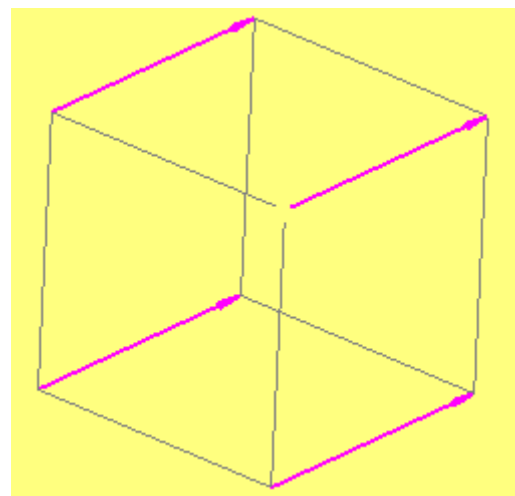


Helper group assists in defining **Reversed Edges** parameter of 1D hypotheses that depend on edge direction.

If **Propagation chains** checkbox is activated, the list is filled with propagation chains found within the model.

When a chain is selected in the list its edges are shown in the Viewer with arrows, which enables choosing a common direction for all chain edges.

Reverse button inverts the common direction of chain edges. If **Add** button is active, some edges of a chain have a different direction, so you can click this button to add them to the list of **Reversed Edges**.

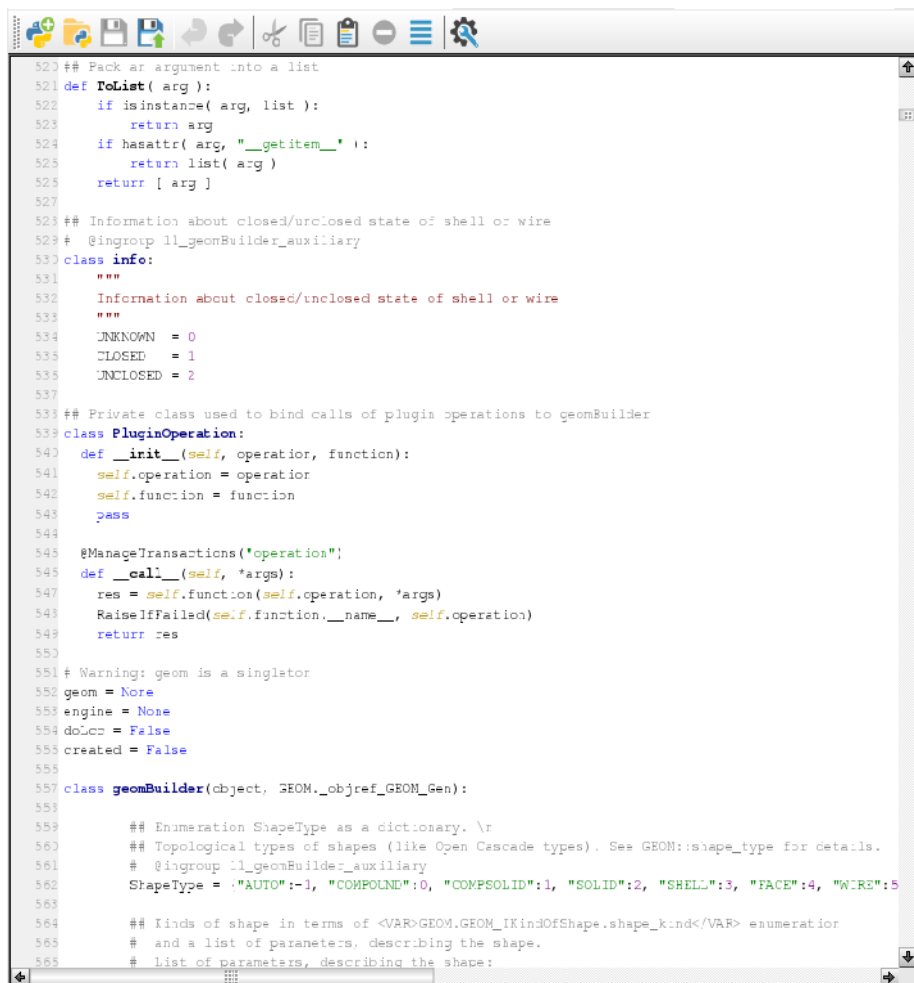


Python editor

An embedded Python editor has been added to the GUI. It can be accessed via the menu **Window** → **New Window** → **Python view**.

The Python editor supports a lot of features:

- Automatic syntax highlighting for Python code;
- Automatic in-place indentation of the code being edited;
- Tabulation management, including visual distinguishing of tabs by the dotted line;
- Display of line number area;
- Display of vertical limiter edge on the right side of the editor;
- Current line highlighting;
- Text wrapping;
- Cursor centering on the scrolling;
- Customizing font properties;
- Management of features via the extended preferences dialog box;
- File operations: new, open, save, save as;
- Undo/redo operations;
- Clipboard operations: cut, copy, paste, clear, select all.



```

520 ## Pack an argument into a list
521 def ToList( arg ):
522     if isinstance( arg, list ):
523         return arg
524     if hasattr( arg, "__getitem__" ):
525         return list( arg )
526     return [ arg ]
527
528 ## Information about closed/unclosed state of shell or wire
529 # @ingroup tl_geomBuilder_auxiliary
530 class info:
531     """
532     Information about closed/unclosed state of shell or wire
533     """
534     UNKNOWN = 0
535     CLOSED = 1
536     UNCLOSED = 2
537
538 ## Private class used to bind calls of plugin operations to geomBuilder
539 class PluginOperation:
540     def __init__(self, operation, function):
541         self.operation = operation
542         self.function = function
543         pass
544
545 @ManageTransactions("operation");
546 def __call__(self, *args):
547     res = self.function(self.operation, *args)
548     RaiseIfFailed(self.function.__name__, self.operation)
549     return res
550
551 # Warning: geom is a singleton
552 geom = None
553 engine = None
554 doLoc = False
555 created = False
556
557 class geomBuilder(object, GEOM._objref_GEOM_Gen):
558
559     ## Enumeration ShapeType as a dictionary. \r
560     ## Topological types of shapes (like Open Cascade types). See GEOM::shape_type for details.
561     # @ingroup tl_geomBuilder_auxiliary
562     ShapeType = {"AUTO":-1, "COMPOLINE":0, "COMPSOLID":1, "SOLID":2, "SHELL":3, "FACE":4, "WIRE":5}
563
564     ## Kinds of shape in terms of <VAR>GEOM.GEOM_IKindOfShape.shape_kind</VAR> enumeration
565     # and a list of parameters, describing the shape.
566     # List of parameters, describing the shape:

```

OTHER IMPROVEMENTS

Geometry module

- New function `TransferData` allows transferring sub-shape names and materials from the initial shape (restored from the external IGES or STEP file) to the repaired one.
- It has become possible to publish results of “Check Free Faces” operation in the study by clicking “Apply” or “Apply and Close” button.
- New option “Hide input objects from the viewer” has been added to the preferences of Geometry module. It provides automatic hiding of objects used as input of Geometry operations and showing only the result shapes.
- “Zooming style” preferences option has been set to the “Relative to the cursor” mode by default.
- Obsolete construction mode “Half-space partition” has been removed from the Partition GUI dialog.
- “What is” dialog box now shows details on the top-level content of a Compound if it is specified as an input object.
- The function `GetNonBlocks` now can optionally check if the contour of a face can be reduced to 4 edges taking into account C1 continuity.
- The color and the texture used for drawing of the clipped data now can be customized in the new “Clipping” group of OCC 3D viewer Preferences.
- New TUI command `GetSubShapeEdgeSorted` allows extracting edges from the argument GEOM object containing edge forming an open wire in a specific order to produce a chain of edges.
- “Filling”, “Glue Faces”, “Glue Edges”, “Remove Internal Faces” and “Sewing” operations now can accept multiple input shapes instead of a compound of shapes.
- “Get Shared Shapes” operation now accepts a compound of input shapes instead of a list.
- “Add point on Edge” operation now allows selecting existing vertices, which will be projected to the chosen edge to produce the intersection.
- It has become possible to create a solid or a compound of solids (compsolid) from a connected set of faces.
- “Detect Self-intersections” operation now allows choosing the level of intersections, e.g. only “edge/edge and lower”, or “face/face and lower”, etc. and publishing the selected results in the study as child objects of the source shape.
- “Shape Process” and other shape healing operations now provide explicit detailed feedback about the modifications applied to the argument shape.
- “Explode” operation now supports additional “Flat” sub-shape type, which is available when processing Compounds at input. If this type is chosen, all top-level “simple” sub-shapes are extracted recursively from the source Compound. Thus the result list of objects will not contain Compounds.
- Pipe generation operations now support automatic creation of groups. It can be activated using check box “Generate Groups”. It is also possible to specify a custom prefix for group names.

Mesh module

- New filtering criterion “Belong to Mesh Group” (`FT_BelongToMeshGroup`) selects elements or nodes included in a specified mesh group. This allows creating “Group on Filter” formed from other groups in the same way as done by “Union Groups”, “Cut Groups” and other operations, but the resulting “Group On Filter” will be automatically updated when reference groups are recomputed.
- “Create group of underlying entities” dialog has been renamed into “Group based of nodes of other groups”. Now it allows creating groups of elements of any dimension as the source group. The old type of behavior is provided by “Include underlying entities only” checkbox.

- “Extrusion”, “Extrusion Along Path” and “Revolution” operations have been improved to sweep nodes, edges and faces at once (previously it was possible to extrude nodes, edges or faces separately).
- The normalized normal of a face is added when "Element info" is called on it.

Kernel module

- Implementation of SALOMESDS: for python global variables in read, read/append modes.

ParaVis module

- PARAVIS 'Gauss Points' filter now generates internal input information for PARAVIS 'Extract Group' filter.

YACS module

- The number of available resources now can be accessed inside `OptimizerLoop`.
- `ForEachLoop` nodes can retrieve passed items even in case of failure.
- `ForEachLoop` nodes are now able to continue to continue as much as possible after an error (new keep going mode of the Executor).

Med module

- Implementation of pickling of instances of the following classes: `dataArrayDouble`, `dataArrayInt`, `MEDCouplingUMesh`, `MEDCouplingCMesh`, `MEDCouplingIMesh`, `MEDCouplingExtrudedMesh`, `MEDCoupling1SGTUMesh`, `MEDCoupling1DGTUMesh`, `MEDCouplingFieldDouble`, `MEDFileUMesh` and `InterpKernelException`. Now instances of these class can be exchanged directly throw YACS python nodes without any additional code. Meshes and fields can be exchanged transparently through processes using multiprocessing standard python package.
- New method `MEDFileUMesh.buildExtrudedMesh` with the same behavior as in Mesh module.
- New methods: `MEDFileUMesh.linearToQuadratic` and `MEDFileUMesh.quadraticToLinear`.
- New method `MEDCouplingPointSet.computeDiameterField` with the same behavior as in Mesh.
- Performance improvement in `MEDCouplingUMesh.buildExtrudedMesh`: decrease of complexity of algorithm ($n*n*p \rightarrow n*p$).
- In `MEDFileFields.LoadSpecificEntities`: improvement of performance for MED file having more than 1000 time steps.
- Behavior change in `MEDCouplingUMesh.buildExtrudedMesh`: `zipCoords` is no more applied.
- Behavior change in `MEDCouplingExtrudedMesh`: constructor does not expect any more that all nodes are fetched.
- New methods: `MEDCouplingPointSet.areAllNodesFetched`, `dataArrayInt.getIdsStrictlyNegative`, `dataArrayInt.replaceOneValByInThis`, `MEDCouplingUMesh.changeOrientationOfCells`, `dataArrayInt.buildUniqueNotSorted`, `dataArrayInt.getMinMaxValues`, `dataArrayInt.getIdentity2`, `MEDFileMesh.rearrangeFamilies`, `MEDFileMesh.checkOrphanFamilyZero`, `MEDCouplingStructuredMesh.getLocationFromCellId`, `MEDCouplingStructuredMesh.getLocationFromNodeId`
- Added Python bindings for `MEDFileUMesh.__getitem__`, `MEDFileUMesh.__setitem__` and `MEDFileUMesh.__delitem__`.

❖ BUG CORRECTIONS

GUI MODULE

22817	<p><i>Summary:</i> EDF GUI: [Regression] Help Browser is not closed when we close SALOME. Help browser is now properly closed when all Salome sessions are closed.</p>
23045	<p><i>Summary:</i> EDF 10641 GUI: Demo plugin is not loaded anymore when closing a study. The procedure of Demo plugin loading has been fixed.</p>

KERNEL MODULE

23059	<p><i>Summary:</i> [CEA 1492] AppQuickStart: invalid translation files. All occurrences of <code>Name_of_Application</code> are now written in square brackets instead of angular to avoid its recognition as XML tag.</p>
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GEOMETRY MODULE

21173	<p><i>Summary:</i> EDF 1787 GEOM: PipeTShape in meter cannot be built. PipeTShape function gives now more stable result thanks to the improvements and bug fixes in Boolean operations and Partition algorithms. A section about model sizes and tolerances has been added to the documentation of Geometry module, in FAQ chapter.</p>
21547	<p><i>Summary:</i> EDF 2194 GEOM: The partition between an ellipsoid and a box gives a bad result. Handling of degenerated parameterization (such as sphere) has been added in <code>IWalker</code>. The fix for this problem has been made in issueOCC25842.</p>
21682	<p><i>Summary:</i> T-shape fails with specific values. An error in <code>MakePipeTShapeFillet</code> functionality has been fixed.</p>
22377	<p><i>Summary:</i> EDF 2752 GEOM: Add a preference in order to automatically unpublished parent objects. New option "Hide input objects from the viewer" has been added to the preferences of Geometry module. It provides automatic hiding of objects used as input of Geometry operations and showing only the result shapes.</p>
22505	<p><i>Summary:</i> EDF 2851 GEOM: Zooming style by default. "Zooming style" preferences option has been set to the "Relative to the cursor" mode by default.</p>

22666	<p>Summary: EDF 7253 GEOM: Add thickness to a shell.</p> <p>It has become possible to add thickness to a shell or a solid.</p> <p>The fix for this problem has been made in issue OCC26046.</p>
22686	<p>Summary: [CEA 1268] Explode a shape into edges sorted in a row from a starting point.</p> <p>New TUI command <code>GetSubShapeEdgeSorted</code> allows extracting edges from the argument GEOM object in a specific order to produce one or more chains of edges.</p>
22745	<p>Summary: [EDF] Improvement of “Sewing” operation.</p> <p>“Sewing” operation now can accept a list of objects instead of one object.</p>
22746	<p>Summary: [EDF] Improvement of “Glue Faces” and “Glue Edges” operations.</p> <p>“Glue Faces” and “Glue Edges” operations have been improved to accept multiple input shapes instead of a compound of shapes.</p>
22747	<p>Summary: [EDF] Improvement of “Get Shared Shapes” operation.</p> <p>“Get Shared Shapes” operation has been improved to accept a compound of input shapes instead of a list.</p>
22748	<p>Summary: [EDF] Improvement of “Filling” operation.</p> <p>“Filling” operation has been improved to accept multiple input shapes instead of a compound of shapes.</p>
22749	<p>Summary: [EDF] Improvement of “Remove Internal Faces” operation.</p> <p>“Remove Internal Faces” operation has been improved to additionally accept a list of input solids instead of only a compound of solids.</p>
22750	<p>Summary: [EDF] Improve ergonomics of the “Detect Self-intersections” dialog.</p> <p>The following changes have been introduced in “Detect Self-intersections” dialog:</p> <ul style="list-style-type: none"> - Computation of self-intersections can be started only by pressing "Compute" button. - “Intersection level” combo-box allow choosing level of intersections, e.g. only “edge/edge and lower”, or “face/face and lower”, etc. - "Apply" button allows publish selected results in the study as child objects of the source shape.
22751	<p>Summary: [EDF] Publish results of the “Check Free Faces” operation.</p> <p>It has become possible to publish results of “Check Free Faces” operation in the study by clicking “Apply” or “Apply and Close” button.</p>
22752	<p>Summary: [EDF] Provide explicit feedback on what has been done by Shape Processing operation.</p> <p>“Shape Process” and other shape healing operations now provide explicit detailed feedback about the modifications applied to the argument shape.</p> <p>The fix for this problem has been made in issues OCC25520 and 25604.</p>

22753	<p><i>Summary:</i> [EDF] Extension of the curves and surfaces.</p> <p>New “Extend” transformation operation allows resizing faces and edges.</p>
22754	<p><i>Summary:</i> [EDF] Surface of a face.</p> <p>New function “Surface from face” extracts the underlying surface of the source face and limits it by the U/V min/max parameters.</p>
22755	<p><i>Summary:</i> [EDF] Improvement of Projection operation.</p> <p>“Projection” dialog has been improved to allow projecting vertices to wires and edges in addition to projection to faces. <code>MakeProjection()</code> python function has been improved correspondingly.</p>
22756	<p><i>Summary:</i> [EDF] Intersection operation</p> <p>Former Section operation has been renamed into Intersection. This operation has been redesigned to allow intersecting different types of shapes and not faces only.</p>
22757	<p><i>Summary:</i> [EDF] Vertex on Edge.</p> <p>“Add point on Edge” operation now allows selecting an existing vertex, which will be projected to the chosen edge to produce the intersection.</p>
22758	<p><i>Summary:</i> [EDF] Creation of a face from a surface and limiting edges.</p> <p>New possibility to build face basing on another existing face has been added in “Create a face” dialog. The resulting face will be built by limiting of the geometrical surface of the source face by the specified wire.</p>
22759	<p><i>Summary:</i> [EDF] Creation of a surface from several edges.</p> <p>It has become possible to create a face from a closed wire and a list of constraints (edges and constraint faces).</p>
22760	<p><i>Summary:</i> [EDF] Improvement of the What Is operation.</p> <p>“What is” dialog box now shows details on the top-level content of a Compound if it is specified as an input object.</p>
22761	<p><i>Summary:</i> [EDF] Improvement of Explode dialog box.</p> <p>“Explode” operation now supports additional “Flat” sub-shape type, which is available when processing Compounds at input. If this type is chosen, all top-level “simple” sub-shapes are extracted recursively from the source Compound. Thus the result list of objects will not contain Compounds.</p>
22763	<p><i>Summary:</i> [EDF] Shape processing.</p> <p>“Shape Processing” now allows detecting small volumes and removing or merging them.</p>
22764	<p><i>Summary:</i> [EDF] Filtering operators in Group and Explode dialog boxes.</p> <p>“Filter” group has been introduced in “Create Group” and “Sub-shapes selection” dialogs to filter out entities with length, area or volume that are less and/or greater than the given value.</p>

22765	<p><i>Summary:</i> [EDF] Improvement of the local selection mechanism</p> <p>The “local” selection capabilities of OCC 3D viewer in the Geometry module now allow selection of different types of entities simultaneously.</p>
22766	<p><i>Summary:</i> [EDF] Transport of names.</p> <p>New function <code>TransferData</code> allows transferring sub-shape names and materials from the initial shape (restored from the external IGES or STEP file) to the repaired one.</p>
22767	<p><i>Summary:</i> [EDF] Construction of composite solids.</p> <p>It has become possible to create a solid from a connected set of faces.</p>
22768	<p><i>Summary:</i> [EDF] Model inspector.</p> <p>New “Inspect Object” operation allows browsing the contents of the selected shape.</p>
22771	<p><i>Summary:</i> [CEA 1311] Creating a smaller wire when using the function "Fuse Collinear Edges Within A Wire".</p> <p>The problem with “Fuse Collinear Edges Within A Wire” algorithm has been fixed.</p>
22775	<p><i>Summary:</i> Add an option on <code>GetNonBlocks</code> to retrieve quadrangular faces defined on C1 edges.</p> <p>The function <code>GetNonBlocks</code> now can optionally check if the contour of a face can be reduced to 4 edges taking into account C1 continuity.</p>
22776	<p><i>Summary:</i> [CEA 1269] Project a wire or a face on a cylinder</p> <p>New operation “Projection on Cylinder” projects a Source planar wire or face on a cylinder defined by its radius. The cylinder’s coordinate system is the same as the global coordinate system. The result is a projection of the source shape wire or a face on the cylinder.</p>
22777	<p><i>Summary:</i> [CEA 1291] Display the name of an object in the 3D View.</p> <p>It has become possible to display the name of a geometrical object in the OCC and VTK 3D viewers. The name label is a 2D text that cannot be scaled or rotated. It is bound to the gravity center of the object.</p>
22781	<p><i>Summary:</i> [CEA 1281] Preferences of colors and textures of clipping planes.</p> <p>The color and the texture used for drawing of the clipped data now can be customized in the new "Clipping" group of OCC 3D viewer Preferences.</p>
22804	<p><i>Summary:</i> [CEA 1332] Import a .brep file with an accent in its name.</p> <p>Problem with Unicode symbols in the file name for Import/Export BREP files has been fixed.</p>
22818	<p><i>Summary:</i> EDF 9341 GUI: A compound of one circle is shown as two superimposing circles in the OCC viewer.</p> <p>The visualization of circle discretization has been improved.</p> <p>The fix for this problem has been made in issue OCC25540.</p>

22822	<p><i>Summary:</i> EDF 9415: Issue with Bezier curve.</p> <p>Handling of cases when the chordal deviation of the displayed edge is much greater than the deflection has been improved.</p> <p>The fix for this problem has been made in issue OCC25841.</p>
22831	<p><i>Summary:</i> Bad order in the OB after "Reduce study".</p> <p>The problem with creation of new objects at the top of the Object Browser after "Reduce Study" has been resolved.</p>
22842	<p><i>Summary:</i> EDF GEOM: [HYDRO] Creating a polyline from a wire in the polyline editor.</p> <p>It has become possible to use an existing wire as a base of polyline creation.</p>
22843	<p><i>Summary:</i> EDF GEOM: [HYDRO] Polyline using a former polyline.</p> <p>The problem with missing last point of an opened polyline created from an existing polyline has been resolved.</p>
22844	<p><i>Summary:</i> EDF GEOM: [HYDRO] Creation of a polyline : modifying the point coordinates.</p> <p>It has become possible to shift and drag and drop points to modify their coordinates during polyline creation.</p>
22845	<p><i>Summary:</i> EDF GEOM: [HYDRO] Hide the coordinate system part of the dialog box in the polyline editor.</p> <p>The system coordinates part of the dialog box has become hidden in the polyline editor.</p>
22846	<p><i>Summary:</i> EDF GEOM: [HYDRO] Creation of a polyline: add intermediate points and remove points.</p> <p>It has become possible to add intermediate points or to remove points during polyline creation.</p>
22849	<p><i>Summary:</i> EDF 9486 GEOM: A partition of 2 shapes stresses a performance issue.</p> <p>The performance of partition algorithm has been improved.</p> <p>The fix for this problem has been made in issue OCC25742.</p>
22851	<p><i>Summary:</i> EDF 9972 GEOM: Regression from 7.4.1 to 7.5.1: MakeCut and MakeCutList of one shape by another gives an unexpected result.</p> <p>The algorithms computing intersection lines on cylinders with two parallel axes, processing trimmed analytical surfaces and checking for wires with derivative angles close to PI have been improved.</p> <p>The fix for this problem has been made in issues OCC25782, OCC25861, OCC25935 and 25969.</p>
22854	<p><i>Summary:</i> [CEA 1420] The Help button on dialog boxes XAO/Export XAO refers to empty file.</p> <p>The Help button in dialogs Import XAO and Export XAO now leads to the correct page.</p>

22857	<p><i>Summary:</i> EDF GEOM 7167: MakeHalfPartition documentation.</p> <p>The construction mode “Half-space partition” has been removed from the Partition GUI dialog.</p>
22858	<p><i>Summary:</i> EDF 10041 GEOM: Regression when importing IGES file taking into account the units.</p> <p>The problem with computation of the center of the view scene for interactive rotation that caused crash of “Fit all” viewer operation has been fixed.</p> <p>The fix for this problem has been made in issue OCC25824.</p>
22861	<p><i>Summary:</i> Export XAO does not export the sub-shapes.</p> <p>The problem with export of sub-shapes in XAO format has been fixed.</p>
22862	<p><i>Summary:</i> EDF GEOM: MakeFillet returns a compound.</p> <p>The output of MakeFillet operation has been corrected.</p>
22865	<p><i>Summary:</i> [CEA 1416] Impossible to visualize the solids found by “Check compound of blocks” in default wireframe mode.</p> <p>The value of “Preview edges width” is now used for display of selected blocks in “Check compound of blocks” operation.</p>
22867	<p><i>Summary:</i> EDF GEOM: Regression with MakeFace.</p> <p>The problem with MakeFace operation has been fixed.</p>
22868	<p><i>Summary:</i> EDF GEOM Regression: Partition between 4 faces does not work in TUI.</p> <p>Partition algorithm has been improved.</p> <p>The fix for this problem has been made in issue OCC25847</p>
22869	<p><i>Summary:</i> EDF 7482 GEOM: Automatically create groups with the generation operations.</p> <p>Pipe generation operations now support automatic creation of groups. It can be activated using check box “Generate Groups”. It is also possible to specify a custom prefix for group names.</p>
22879	<p><i>Summary:</i> [CEA 1426] TestMeasureOperations fails.</p> <p>The problem with GEOM_TestMeasures.py has been fixed by adding a conditional check for SALOME version.</p>
22883	<p><i>Summary:</i> EDF GEOM: Regression with GetShapesOnQuadrangle.</p> <p>A regression in GetShapesOnQuadrangle function has been fixed.</p>
22885	<p><i>Summary:</i> EDF 10392 HYDRO + GEOM: Spline of a polyline is not possible and returns an error.</p> <p>It has become possible to build a spline by interpolating a polyline.</p>

22889	<p><i>Summary:</i> GetSame does not return the right shape.</p> <p>The formulas for calculation of hyperbola have been improved.</p> <p>The fix for this problem has been made in issue OCC26041.</p>
22934	<p><i>Summary:</i> EDF GEOM Regression: MakePartition returns a wrong shape.</p> <p>Processing of overlapping curves has been improved.</p> <p>The fix for this problem has been made in issues OCC25951 and OCC26080.</p>
23034	<p><i>Summary:</i> Information after a transformation without copy.</p> <p>Information window in GEOM module has been improved to show information on both creation and all transformation operations of a GEOM object.</p>
23038	<p><i>Summary:</i> EDF GEOM Regression: SIGSEGV error with MakePipeWithDifferentSections.</p> <p>The problem with construction of a pipe with different sections has been fixed.</p>
23043	<p><i>Summary:</i> EDF 10596 GEOM: GetShapesOnShape does not return all shapes expected.</p> <p>The algorithm creating groups using GetShapesOnShape function has been fixed.</p>
23044	<p><i>Summary:</i> EDF GEOM Regression: Fatal error when using 3D Sketch.</p> <p>Handling of invalid bounding boxes has been improved.</p> <p>The fix for this problem has been made in issues OCC26076 and OCC26120.</p>
23046	<p><i>Summary:</i> EDF SMESH Regression: Field "Compare" in filters is not updated anymore.</p> <p>The problem with an empty field of "Belong to Geom" criterion has been fixed.</p>
23048	<p><i>Summary:</i> [CEA 1474] Performance regression on MakeGlueEdges.</p> <p>The algorithm of curve on surface projection has been significantly accelerated.</p>
23055	<p><i>Summary:</i> EDF 10725 GEOM: In the shape recognition functionality, it is not possible to select an area for the filtering sample.</p> <p>The problem with mirroring of an image imported in the viewer has been resolved.</p>
23057	<p><i>Summary:</i> EDF GEOM: Direction of edge of a circle has changed on Salome V7_6_BR.</p> <p>Fixed regressions with disk creation operation.</p>
23058	<p><i>Summary:</i> [CEA 1493] Regression: MakeDiskPntVecR does not create the same disk as before.</p> <p>Fixed regressions with disk creation operation.</p>
23063	<p><i>Summary:</i> EDF GEOM Regression: Partition with tool returns a bad result.</p> <p>The algorithm creating pave blocks from intersection curves has been improved.</p> <p>The fix for this problem has been made in issue OCC26218.</p>

23095	<p><i>Summary:</i> EDF GEOM Regression: Common returns wrong result with Salome V7_6_BR/OCCT 6.9.0.</p> <p>Processing of curves to be reversed has been modified taking into account reversed parameter.</p> <p>The fix for this problem has been made in issue OCC26224.</p>
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MESH MODULE

22264	<p><i>Summary:</i> EDF 2648 GEOM: Propagate edges automatic orientation.</p> <p>“Helper” group has been added in dialogs to assists in defining “Reversed Edges” parameter for 1D hypotheses, which depend on edge direction.</p>
22599	<p><i>Summary:</i> EDF 8159 SMESH: Bad groups created by extrusion.</p> <p>Naming of extruded groups has been corrected.</p>
22635	<p><i>Summary:</i> EDF 8345 - Creation of group based on groups.</p> <p>“Create group of underlying entities” dialog has been renamed into “Group based of nodes of other groups”. Its functionality has been extended and now it can create groups of elements of the any dimension as the source group. The old type of behavior is provided by “Include underlying entities only” checkbox.</p>
22649	<p><i>Summary:</i> EDF SMESH: Another Origin and Base Vector are created when we assign a set of hypotheses in SMESH.</p> <p>The problem with duplication of origin and base vectors at the first activation of Mesh menu in Mesh module has been fixed.</p>
22792	<p><i>Summary:</i> EDF 8159 SMESH: Multi-dimensional extrusion/extrusion along a path/revolution.</p> <p>“Extrusion”, “Extrusion Along Path” and “Revolution” operations have been improved to sweep nodes, edges and faces at once (previously it was possible to extrude nodes, edges or faces separately).</p>
22833	<p><i>Summary:</i> [CEA 1346] Extrude a group of faces following the normal of each face.</p> <p>Extrusion along average normal of faces sharing a node has been implemented.</p>
22834	<p><i>Summary:</i> [CEA 1347] Viscous layers: be able to choose the extrusion method.</p> <p>Two new extrusion methods “Face Offset” and “Node Offset” now can be used at viscous layers construction. The extrusion method defines how positions of new nodes are found during prism construction and how creation of distorted and intersecting prisms is prevented.</p>
22839	<p><i>Summary:</i> EDF 9647 SMESH: CreateGroupFromGEOM does not return all elements.</p> <p>Regression of MG-Tetra plug-in, consisting in invalid binding of tetrahedra to solids in case of quadrangle 2D mesh, has been fixed.</p>
22860	<p><i>Summary:</i> EDF 9944 SMESH: Regression when scaling a mesh.</p> <p>Regression of anisotropic mesh scaling has been fixed.</p>

22874	<p><i>Summary:</i> [CEA 1425] Performance SMESH Module. Some memory leaks in Mesh module have been eliminated.</p>
22876	<p><i>Summary:</i> EDF 8425 SMESH: Get the normal of a face in the GUI. The normalized normal of a face is added when "Element info" is called on it.</p>
22877	<p><i>Summary:</i> EDF 10054 SMESH: Add a new filter <code>BelongToSmeshGroup</code> for Create Group. New filtering criterion "Belong to Mesh Group" (<code>FT_BelongToMeshGroup</code>) allows selecting elements or nodes included in a specified mesh group.</p>
22878	<p><i>Summary:</i> [CEA 1424] Performances of Compute method from Mesh class in <code>smeshBuilder.py</code>. Additional Boolean parameter <code>refresh</code> (false by default) has been added to <code>Compute</code>, <code>Clear</code> and <code>ClearSubMesh</code> methods.</p>
23023	<p><i>Summary:</i> [CEA 1445] Missing <code>ExtrusionAlongPathObject</code> function. Some missing <code>SMESH_MeshEditor</code> methods have been restored.</p>
23024	<p><i>Summary:</i> [CEA 1447] Regression <code>imp_1346_CAS2.py</code>. A regression of <code>ExtrusionByNormal()</code> method has been fixed.</p>
23027	<p><i>Summary:</i> EDF SMESH Regression: Compute fails with MG-Tetra/MG-CADSurf on a partition. The problem with wrong Subdomain assignment in output files of MG-Tetra has been fixed.</p>
23028	<p><i>Summary:</i> EDF 10508 SMESH: <code>GroupOnFilter</code> returns an empty group when the geometry is not published. The problem with an empty group on filter if created basing on a non-published geometry has been fixed.</p>
23032	<p><i>Summary:</i> EDF SMESH: Projection 1D-2D fails with Netgen 1D-2D. Projection 1D-2D algorithm has been improved to correctly process the case of projection to a cylindrical surface with holes.</p>
23033	<p><i>Summary:</i> EDF 10568 SMESH: wrong header when exporting 2D to gmf. GMF now uses version 2 instead of 3, because the newer version is not supported by <code>MeshGems</code> meshers.</p>
23036	<p><i>Summary:</i> [CEA 1459] Regression projection 1D2D. A regression has been fixed in <code>Projection1D/2D</code> algorithm.</p>
23047	<p><i>Summary:</i> [CEA 1472] Incorrect mesh with Netgen 1D-2D-3D but not reported in error. The problem with meshing failure of NETGEN 1D-2D-3D algorithm on a coarse quadrangular 2D mesh has been fixed.</p>

23049	<p><i>Summary:</i> [CEA 1475] Increase the size of some dialog boxes.</p> <p>The default dimensions of dialogs for some hypotheses boxes have been corrected.</p>
23050	<p><i>Summary:</i> EDF 10631 SMESH: Nodes outside the plane after <code>ConvertToQuadratic</code></p> <p>The problem with Convert To Quadratic operation, which moved some boundary nodes off the surface, has been fixed.</p>
23051	<p><i>Summary:</i> [CEA 1470] SMESH NETGEN - Local refinement not computed and dumped properly.</p> <p><code>NETGEN_Parameters.SetLocalSizeOnShape()</code> has been protected against non-published shapes.</p>
23056	<p><i>Summary:</i> EDF SMESH Regression: Projection fails.</p> <p>Regression of Projection 2D algorithm has been fixed.</p>
23061	<p><i>Summary:</i> [CEA 1488] Import 1D-2D fails sometimes in relation with the source face discretization.</p> <p>An error message issued by "Use Existing 2D Elements" algorithm in case of impossibility to create segments on a seam edge has been corrected.</p>
23092	<p><i>Summary:</i> EDF 10836 SMESH: <code>UseExisting2DElements</code> fails when geometry contains more than one face.</p> <p>"Use Existing 2D Elements" algorithm now produces complete meshing.</p>

PARAVIS MODULE

22884	<p><i>Summary:</i> EDF 9622 MED: Filter pipeline Gauss Points + Extract Group is not working.</p> <p>PARAVIS 'Gauss Points' filter now generates internal input information for ParaVis 'Extract Group' filter. It resolves the problem with empty result of 'MedReader → GaussPoints → ExtractGroup' pipeline.</p>
22795	<p><i>Summary:</i> [CEA 1324] Display panel is blank</p> <p>The fix has been integrated to ParaView version 4.3.1.</p>
22921	<p><i>Summary:</i> [CEA 1438] Remove the checkbox in the "time steps"</p> <p>Implemented a workaround for ParaView problem.</p>
23021	<p><i>Summary:</i> [CEA 1440] Setting the field to display in python with reader. <code>AllArrays</code> does not change the fields set in the <code>MEDReader</code> properties.</p> <p>The problem with Python script has been fixed.</p>
23022	<p><i>Summary:</i> [CEA 1442] SSH: Movement of mesh jerky</p> <p>The problem is solved by adding <code>-DVTK_REPORT_OPENGL_ERRORS:BOOL=OFF</code> option at the ParaView configuration step.</p>

23025	<p><i>Summary:</i> [CEA 1444] In built-in server mode, the mouse becomes a hourglass pointer while rotating the view.</p> <p>A call-back in PARAVIS module that showed wait cursor on any VTK event has been removed.</p>
23037	<p><i>Summary:</i> [CEA 1458] The dump python does not save the camera position.</p> <p>The Python tracing mechanism has been fixed.</p>
23041	<p><i>Summary:</i> [CEA 1468] In master with ParaView 4.3.1, the trace does not appear in the dump.</p> <p>GUI of PARAVIS module has been re-implemented following "light" approach.</p>
23096	<p><i>Summary:</i> [CEA 1518] load python script: <code>ServerModifTime</code> does not exist.</p> <p>The variable <code>_is_fields_status_changed</code> has been removed to avoid problems in multi-clients mode.</p>

YACS MODULE

22323	<p><i>Summary:</i> [CEA 914] YACS: quantity of resources in <code>OptimizerLoop</code>.</p> <p>The number of available resources now can be accessed inside <code>OptimizerLoop</code>.</p>
N/A	<p><i>Summary:</i> Bug correction for list of strings coming from a <code>PyNode</code> to a <code>ForEachLoop</code></p>

BLSURF (MG-CADSURF) PLUGIN MODULE

21894	<p><i>Summary:</i> [CEA 683] Salome fails with proximity on spheres multi-layers not collapsed.</p> <p>The problem with crash at mesh generation has been fixed.</p>
22903	<p><i>Summary:</i> EDF 10469 BLSURFPLUGIN: <code>setSizeMap</code> is not taken into account when we add <code>AddPreCadFacesPeriodicity</code> parameter.</p> <p>Size Map is now properly taken into account if Periodicity is set.</p>
23026	<p><i>Summary:</i> [CEA 1411] Problems with MeshGems 2.0.</p> <p>The problem with MeshGems 2.0 has been fixed.</p>
23035	<p><i>Summary:</i> EDF 10591 BLSURFPLUGIN: A dump python script is not valid when reloaded after creation.</p> <p>The problem with of Python Dump of <code>MG_CADSurf_Parameters.SetAttractorGeom()</code> has been fixed.</p>

HEXOTIC (MG-HEXA) PLUGIN MODULE

22515	<p><i>Summary:</i> [CEA 1074] Hexotic does not work under Windows 7 64 bits.</p> <p>Hexotic plugin now works under Windows 7 64 bits.</p>
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MED MODULE

N/A	<i>Summary:</i> MEDFileField1TS.setFieldProfile on a false profile for structured mesh
N/A	<i>Summary:</i> Invalid python wrapping for MEDFileMesh.setGroupsAtLevel (for structured mesh users)

OTHER ISSUES

22778	<i>Summary:</i> [CEA 1298] Python file compilation should be done at build time, not install time. The installation time of Python files has been greatly reduced by exclusion of their unconditional byte-compilation, which is currently performed depending on comparison of file timestamps.
23090	<i>Summary:</i> [CEA 1510] Error at Salome launch with Japanese language. Fixed problem related to the usage of Unicode in SALOME.

❖ OCCT 6.9.0 BUG CORRECTIONS

This chapter lists bug corrections and improvements made for SALOME project in Open CASCADE Technology. These bug corrections and improvements are included into OCCT version 6.9.0.

Note that only the issues related to SALOME platform are listed below. For complete list of changes introduced by Open CASCADE Technology version 6.9.0 please refer to its Release Notes: <http://www.opencascade.com/news/issue217/>.

CORRECTIONS INCLUDED INTO OFFICIAL OCCT VERSION 6.9.0

OCC25193	<i>Summary:</i> Bad Intersection curve obtained by Surface/Surface Intersection Algorithm. Referenced by 0052446: Partition is failed
OCC25271	<i>Summary:</i> Wrong result obtained by solid classifier algorithm Referenced by 0022702: [CEA 1292] MinDistance between 2 polyhedrons that are not intersecting returns 0
OCC25354	<i>Summary:</i> Intersection operation Referenced by 0022756: [EDF] Intersection operation
OCC25398	<i>Summary:</i> Modeling Algorithms - Provide shape proximity detector Referenced by 0022762: [EDF] Fast detection of face/face face/solid solid/solid interference
OCC25432	<i>Summary:</i> Wrong result obtained by MakerVolume operator. Referenced by 0021836: EDF 2204 GEOM : Regression in partition operation
OCC25436	<i>Summary:</i> Visualization - AIS_InteractiveContext - HighlightPreviousDetected() should switch from first <i>Summary:</i> value in the list to the last Referenced by 0022765: [EDF] Improvement of local selection mechanism
OCC25449	<i>Summary:</i> Excess vertex in result of General Fuse operation. Referenced by 0021836: EDF 2204 GEOM : Regression in partition operation
OCC25450	<i>Summary:</i> Common operation returns wrong shape Referenced by 0022743: EDF GEOM: Regression in MakePipeTShapeChamfer: Some faces are missing in a GetShapesOnCylinder result
OCC25455	<i>Summary:</i> fixshape works at the second attempt Referenced by 0022743: EDF GEOM: Regression in MakePipeTShapeChamfer: Some faces are missing in a GetShapesOnCylinder result
OCC25456	<i>Summary:</i> BOPAlgo_CheckerSI reports an error on the given shape Referenced by 0022689: EDF GEOM: Regression - Extrusion along a path with a divided disk

OCC25465	<p><i>Summary:</i> Excess vertex in the result of CUT operation This regression has been detected by SALOME non-regression test.</p>
OCC25480	<p><i>Summary:</i> Incorrect result of BRepOffsetAPI_MakePipe Referenced by 0022697: EDF 8788 GEOM: Regression with MakePipe</p>
OCC25488	<p><i>Summary:</i> Wrong result of two trimmed cylinders intersection This regression has been detected by SALOME non-regression test.</p>
OCC25494	<p><i>Summary:</i> Wrong result obtained by projection algorithm Referenced by 0052545: Glue Faces operation creates an invalid shape</p>
OCC25505	<p><i>Summary:</i> General Fuse produces self-intersection shape This regression has been detected by SALOME non-regression test.</p>
OCC25509	<p><i>Summary:</i> Wrong shape considered as valid by checkshape Referenced by OCC25505: General Fuse produces self-intersection shape</p>
OCC25520	<p><i>Summary:</i> To provide info on what has been done by Shape Process and Shape Fix. Referenced by 0022752: [EDF] Provide explicit feedback on what has been done by Shape Processing operation.</p>
OCC25523	<p><i>Summary:</i> Fail to read back solid written to STEP This regression has been detected by SALOME non-regression test.</p>
OCC25529	<p><i>Summary:</i> ShapeProcessAPI - introduce DropSmallSolids operator Referenced by 0022763: [EDF] Shape processing</p>
OCC25540	<p><i>Summary:</i> Visualization - discretization of the circle differs in shaded and wireframe modes. Referenced by 0022818: EDF 9341 GUI: A compound of one circle is shown as two superimposing circles in the OCC viewer.</p>
OCC25559	<p><i>Summary:</i> SIGSEGV in TKMath when computing max tolerance of curve on surface Referenced by 0022763: [EDF] Shape processing</p>
OCC25584	<p><i>Summary:</i> Wrong result obtained by PerformInfinitePoint Test Referenced by:</p> <ul style="list-style-type: none"> - OCC25505: General Fuse produces self-intersection shape - OCC25509: Wrong shape considered as valid by checkshape

OCC25592	<p><i>Summary:</i> Bad result of Fillet operation</p> <p>Referenced by 0022706: EDF GEOM: Regression in T-shape pipe primitive with fillet: HexMesh option induces an error</p>
OCC25597	<p><i>Summary:</i> Invalid curve on surface in the result of General Fuse operation</p> <p>0022706: EDF GEOM: Regression in T-shape pipe primitive with fillet: HexMesh option induces an error</p>
OCC25600	<p><i>Summary:</i> Wrong result of Boolean FUSE operation</p> <p>This regression has been detected by SALOME non-regression test.</p>
OCC25604	<p><i>Summary:</i> To provide info on what has been done by DropSmallSolids operator</p> <p>Referenced by;</p> <ul style="list-style-type: none"> - 0022752: [EDF] Provide explicit feedback on what has been done by Shape Processing operation - 0022763: [EDF] Shape processing
OCC25614	<p><i>Summary:</i> Provide API access to the new functionalities of Boolean Components</p> <p>Improve BOP API to process multiple arguments</p>
OCC25657	<p><i>Summary:</i> Bad result of Fillet operation</p> <p>Referenced by 0022706: EDF GEOM: Regression in T-shape pipe primitive with fillet: HexMesh option induces an error</p>
OCC25670	<p><i>Summary:</i> Incorrect sprops/vprops/lprops check in heal drop_small_solids test cases</p> <p>This regression has been detected by SALOME non-regression test.</p>
OCC25677	<p><i>Summary:</i> Wrong intersection curves between faces</p> <p>Referenced by 0022706: EDF GEOM: Regression in T-shape pipe primitive with fillet: HexMesh option induces an error</p>
OCC25701	<p><i>Summary:</i> Problem with the symmetry of fillet on two perpendicular cylinders</p> <p>Referenced by 0022706: EDF GEOM: Regression in T-shape pipe primitive with fillet: HexMesh option induces an error</p>
OCC25715	<p><i>Summary:</i> Intersection between cylinders produces excess vertices</p> <p>This regression has been detected by SALOME non-regression test.</p>
OCC25721	<p><i>Summary:</i> Wrong result obtained by Common operator.</p> <p>Referenced by 0052601: [TC7.5.1] Bad result of Partition</p>
OCC25735	<p><i>Summary:</i> Wrong solid is considered as valid by checkshape</p> <p>Referenced by 0052600: [TC7.5.1] Partition failure: tool shape is kept in the result</p>

OCC25742	<p><i>Summary:</i> A partition of 2 shapes stresses a performance issue.</p> <p>Referenced by 0022849: EDF 9486 GEOM: A partition of 2 shapes stresses a performance issue.</p>
OCC25743	<p><i>Summary:</i> Add FixMode parameter to DropSmallSolids operator</p> <p>Referenced by 0022763: [EDF] Shape processing</p>
OCC25782	<p><i>Summary:</i> The result of intersection between two cylinders is incorrect.</p> <p>Referenced by 0022851: EDF 9972 GEOM: Regression from 7.4.1 to 7.5.1: MakeCut and MakeCutList of one shape by another gives an unexpected result.</p>
OCC25809	<p><i>Summary:</i> Visualization, TKOpenGL - fix texture mapping in capping</p> <p>This regression has been detected by SALOME non-regression test.</p>
OCC25814	<p><i>Summary:</i> Visualization, Prs3d_WFShape - AddPolygon() - always use polygonal representation from edge regardless from requested deflection</p> <p>This regression has been detected by SALOME non-regression test.</p>
OCC25824	<p><i>Summary:</i> Visualization, TKV3d - skip infinite presentations when computing gravity center of the view scene.</p> <p>Referenced by 0022858: EDF 10041 GEOM: Regression when importing IGES file taking into account the units.</p>
OCC25838	<p><i>Summary:</i> Wrong result obtained by General Fuse operator.</p> <p>Referenced by 0052600: [TC7.5.1] Partition failure: tool shape is kept in the result</p>
OCC25841	<p><i>Summary:</i> Incorrect edge displaying.</p> <p>0022822: EDF 9415: Issue with Bezier curve</p>
OCC25842	<p><i>Summary:</i> Wrong intersection 2D-curves obtained for pair of faces.</p> <p>Referenced by 0021547: EDF 2194 GEOM: The partition between an ellipsoid and a box gives a bad result.</p>
OCC25847	<p><i>Summary:</i> Wrong result obtained by General Fuse operator.</p> <p>Referenced by 0022868: EDF GEOM Regression: Partition between 4 faces does not work in TUI.</p>
OCC25861	<p><i>Summary:</i> Wrong result obtained by projection algorithm.</p> <p>Referenced by 0022851: EDF 9972 GEOM: Regression from 7.4.1 to 7.5.1: MakeCut and MakeCutList of one shape by another gives an unexpected result.</p>
OCC25883	<p><i>Summary:</i> BRepOffsetAPI_MakePipeShell produces invalid result</p> <p>Referenced by 0022870: EDF 10111 GEOM: MakePipe of a wire along an edge fails.</p>

OCC25887	<p><i>Summary:</i> Invalid pipe construction</p> <p>Referenced by 0022881: EDF 10225 GEOM: MakePipe produces an invalid shapes.</p>
OCC25897	<p><i>Summary:</i> Visualization, TKOpenGL - disable FBO blitting after first failure on broken OpenGL context</p> <p>This regression has been detected by SALOME non-regression test.</p>
OCC25932	<p><i>Summary:</i> Visualization - method AIS_Shape - setWidth() should set line width for FaceBoundaryAspect</p> <p>This regression has been detected by SALOME non-regression test.</p>
OCC25935	<p><i>Summary:</i> Visualization, TKV3d, Exception when displaying shell in the viewer.</p> <p>Referenced by 0022851: EDF 9972 GEOM: Regression from 7.4.1 to 7.5.1: MakeCut and MakeCutList of one shape by another gives an unexpected result.</p>
OCC25951	<p><i>Summary:</i> Exception in intersection operation.</p> <p>Referenced by 0022934: EDF GEOM Regression: MakePartition returns a wrong shape.</p>
OCC25969	<p><i>Summary:</i> Wrong result obtained by 2d classifier algorithm.</p> <p>Referenced by 0022851: EDF 9972 GEOM: Regression from 7.4.1 to 7.5.1: MakeCut and MakeCutList of one shape by another gives an unexpected result.</p>
OCC25974	<p><i>Summary:</i> Visualization - fix misprint in method naming of Graphic3d_GraduatedTrihedron</p> <p>This regression has been detected by SALOME non-regression test.</p>
OCC25990	<p><i>Summary:</i> Visualization - result of compilation depends on the order of included OCCT header files</p> <p>This regression has been detected by SALOME non-regression test.</p>
OCC26008	<p><i>Summary:</i> Wrong intersection 2D-curves obtained for pair of faces.</p> <p>Referenced by 0022631: [CEA 1204] MakePartition does not work on a compound made of hollowed spheres</p>
OCC26025	<p><i>Summary:</i> Visualization, TKOpenGL - stereoscopic output does not work</p> <p>This regression has been detected by SALOME non-regression test.</p>
OCC26041	<p><i>Summary:</i> GeomLib_Tool::Parameter method fails.</p> <p>Referenced by 0022889: GetSame does not return the right shape.</p>
OCC26076	<p><i>Summary:</i> Visualization - empty bounding box of a shape after closing local context.</p> <p>Referenced by 0023044: EDF GEOM Regression: Fatal error when using 3D Sketch.</p>

OCC26080	<p><i>Summary:</i> Wrong result obtained by General Fuse operator.</p> <p>Referenced by 0022934: EDF GEOM Regression: MakePartition returns a wrong shape.</p>
OCC26109	<p><i>Summary:</i> Visualization - add ChangeAxisAspect(int) method to Graphic3d_GraduatedTrihedron</p> <p>This regression has been detected by SALOME non-regression test.</p>
OCC26120	<p><i>Summary:</i> Visualization - segmentation fault in AIS_Selection.</p> <p>Referenced by 0023044: EDF GEOM Regression: Fatal error when using 3D Sketch.</p>
OCC26130	<p><i>Summary:</i> PCurve creation failure</p> <p>Referenced by 0052707: MakePipeTShapeFillet is failed</p>
OCC26147	<p><i>Summary:</i> Visualization - restore the ability to pick only fully included objects in rectangular selection</p> <p>This regression has been detected by SALOME non-regression test.</p>
OCC26152	<p><i>Summary:</i> Wrong result obtained by projection algorithm.</p> <p>Referenced by 0052707: MakePipeTShapeFillet is failed</p>
OCC26159	<p><i>Summary:</i> Visualization - revise tolerance implementation for selection</p> <p>This regression has been detected by SALOME non-regression test.</p>

CORRECTIONS INCLUDED INTO PATCH FOR OCCT VERSION 6.9.0

OCC26218	<p><i>Summary:</i> Wrong result done by General Fuse algorithm.</p> <p>Referenced by 0023063: EDF GEOM Regression: Partition with tool returns a bad result.</p>
OCC26224	<p><i>Summary:</i> Wrong result obtained by Common operator.</p> <p>Referenced by 0023095: EDF GEOM Regression: Common returns wrong result with Salome V7_6_BR/OCCT 6.9.0.</p>

❖ SUPPORTED DISTRIBUTIONS AND PRE-REQUISITES

SALOME is a cross-platform solution that supports Linux and Windows. It is distributed as open-source software under the terms of the GNU LGPL license.

SALOME 7.6.0 comes with the same versions of pre-requisites on all supported platforms (with some minor exceptions). The table below lists the versions of the pre-requisite products used by SALOME platform. Other versions of the products can also work but it is not guaranteed.

Product	Version	GUI (APP)	KERNEL	GEOM	SMESH	MED	YACS	PARAVIS	HOMARD	HEXABLOCK	NETGENPLUGIN	GHS3DPLUGIN	GHS3DPRRPLUGIN	BLSURFPLUGIN	HexoticPLUGIN	HEXABLOCKPLUGIN	HYBRIDPLUGIN
gcc*	4.1***	X	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
Microsoft Visual C++**	2010	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X	X
cmake	2.8.10.2	X	X	X	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	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	6.9.0	X		X	X	X	X	X	X	X	X	X	X	X	X	X	X
Qwt	6.1.0	X			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	3.0.8p1				X	X		X	X		X		X				
Vtk	6.2	X		X	X	X		X		X	X	X	X	X	X	X	X
numpy	1.8.2		X														
lapack	3.5.0		X														
graphviz	2.38.0	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						
Metis	4.0					X											
Scotch	5.1.11					X											
libxml2	2.9.0	X	X			X	X										
Distene MeshGems	2.0-10											X	X	X	X		X
Sphinx	1.1.3		X	X	X		X		X	X							
libBatch	2.3.0		X														
Cgns	3.1.3				X												
ParaView	4.3.1	X						X									
Homard	11.1								X								
simanio	1.0		X														

*) Not included into SALOME distribution, Linux only
 **) Not included into SALOME distribution, Windows only
 ***) 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
GNU make*	3.80***	X	X	X	X	X	X	X	X	X	X	X	X		X
Microsoft Visual C++**	2010	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	6.9.0		X		X	X	X	X		X		X			
Qwt	6.1.0				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	3.0.8p1		X	X	X	X									
Vtk	6.2		X		X			X	X	X		X			
graphviz	2.38.0	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 distribution, Linux only
 **) Not included into SALOME distribution, Windows only
 ***) 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	Optional
tk	8.6.0	Open CASCADE Technology	Optional
tclX	8.4.1	Open CASCADE Technology	Optional
Jinja2	2.6	Sphinx	
pygments	1.5	Sphinx	
setuptools	0.6c11	Sphinx	
docutils	0.10	Sphinx	
freetype	2.4.11	Open CASCADE Technology	
freeimage	3.16.0	Open CASCADE Technology	Optional
gl2ps	1.3.8	Open CASCADE Technology, VTK	Optional
Intel TBB	4.2.4	Open CASCADE Technology	Optional
xdata	0.9.9		Can be used to create 3 rd - party SALOME modules
wso2-wsf-cpp	2.1.0	SIMANIO	Optional
matplotlib	1.4.3	ParaView	Optional
distribute	0.6.28	matplotlib	
py-parsing	2.0.3	matplotlib	
python-dateutil	2.3	matplotlib	
pytz	2015.2	matplotlib	
scipy	0.14.1	matplotlib	
six	1.8.0	matplotlib	

SALOME 7.6.0 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.

Software Requirements

Product	Compilation and Development		Execution		Remarks
	Mandatory	Optional	Mandatory	Optional	
gcc	X		X		
GNU make	X				
Microsoft Visual C++	X		X		For execution, runtime libraries are only required
cmake	X				
Python	X		X		
Qt	X		X		
sip	X				
PyQt	X		X		
Boost	X		X		
Swig	X				
OCCT	X		X		
Qwt	X		X		
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
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		
MeshGems	X	X	X		Compilation: mandatory for BLSURFPLUGIN only, optional for HEXOTIC plugin Runtime: mandatory for BLSURFPLUGIN, GHS3DPLUGIN, GHS3DPRLPLUGIN, HexoticPLUGIN, HYBRIDPLUGIN
Sphinx		X			
libBatch		X		X	Required only if used at compilation step
ParaView	X		X		Mandatory for PARAVIS module; optional for GUI module
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
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
simanio		X		X	Required only if used at compilation step
wso2-wsf-cpp		X		X	Required only if KERNEL is built with SIMAN support
matplotlib				X	
scipy				X	

❖ SALOME SYSTEM REQUIREMENTS

Minimal Configuration:

- Processor: Pentium IV
- 512 MB RAM
- Hard Drive Space: 3 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 7.6.0 can be retrieved from the sources repositories using V7_6_0 tag; the complete list of repositories can be found at <https://git.salome-platform.org/gitweb/>.

SALOME version 7.6.0 uses patches for some third-party pre-requisite products, such as Open CASCADE Technology, ParaView, Netgen and other. These patches solve different problems detected within SALOME project.

All pre-requisites can be obtained either from the Linux distribution (please be sure to use a compatible version) in form of native package or from the distributors of these pre-requisites.

❖ LICENSE

SALOME platform is distributed under terms of the GNU Lesser General Public License (LGPL) license version 2.1. All used pre-requisites use similar or compatible licenses (with minor exceptions). Detail information about licenses used by SALOME and its pre-requisites can be found on the following page: <http://www.salome-platform.org/downloads/license/>.

❖ KNOWN PROBLEMS AND LIMITATIONS

- The following modules are obsolete and not included into SALOME 7.6 release: FILTER, SUPERV, MULTIPR, VISU (Post-Pro). 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 cannot 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 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 and deprecation of Post-Pro module 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 OCCT 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
```
- 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 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 64-bit platforms when building ParaMEDCorba 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
```
 - Loading big files in ParaVis might render SALOME instable. This will be fixed in the next release and can be avoided in the current version by one of the two solutions below:
 - In ParaVis settings (ParaVis tab), disable the use of the external pvserver. This approach has the limitation that it is not possible to execute ParaVis' Python scripts outside the SALOME graphical interface (for instance, from an external terminal).
 - In ParaVis settings (ParaView tab → RenderView tab), increase the amount of memory under "Remote/Parallel rendering options" to something bigger than the default 20 MB (for example 200 MB).
 - ParaVis module executes ParaView-related code in the standalone pvserver process that is launched with `--offscreen-rendering` option; this can cause problems with displaying data in ParaVis module if graphic card driver does not support off-screen rendering feature.