

# Open CASCADE 5.0.1 Maintenance release

## Release Notes

### Overview

Open CASCADE 5.0.1 is a maintenance release, which includes bug fixes, new features and improvements over major release 5.0.

Version 5.0.1 is binary incompatible with Version 5.0, so applications linked against the latter must be recompiled to run with 5.0.1

### Table of Contents

- **New Features**
  - ⇒ [Data Exchange](#)
  - ⇒ [Modeling Algorithms](#)
  - ⇒ [Modeling Data](#)
  - ⇒ [Visualization](#)
  - ⇒ [Test Harness](#)
  
- **Improvements**
  - ⇒ [Application Framework](#)
  - ⇒ [Data Exchange](#)
  - ⇒ [Modeling Algorithms](#)
  - ⇒ [Modeling Data](#)
  - ⇒ [Visualization](#)
  - ⇒ [Test Harness](#)
  - ⇒ [Rebuilding Modules](#)
  
- **Bug Fixes**



## New Features

### Data Exchange

- **ShapeUpgrade package**

A new ShapeUpgrade\_RemoveLocations class has been added to the ShapeUpgrade package. This class is intended to remove all locations from sub-shapes starting with the type specified by TopAbs\_ShapeEnum in the method SetRemoveLevel(). With the default value of this parameter locations will be removed from all sub-shapes of a shape except for compounds.

### Modeling Algorithms

- New BOPTools\_Tools3D::DoSplitSEAMOnFace() method with 4 arguments has been added to the BOPTools package. This method is intended to make a seam edge for a given face based on U-periodic and/or V-periodic surface. This seam edge is made from a given edge by adding pcurves thereto.

*Note : the old method BOPTools\_Tools3D::DoSplitSEAMOnFace() with 2 arguments treats only faces based on U-periodic surfaces.*

- A history mechanism has been added to New Boolean Operations with a view to using it in the Topological Naming algorithm of OCAF.

*Note : Topological Naming for some standard modeling operations (Boolean operations, filleting, pipes, etc) is provided in the BRepNaming package, which is an optional component provided on top of Open CASCADE. The specific BRepNaming algorithm is intended to collect history information in the OCAF document, which can be used for tracking shape modification.*

To support the history mechanism new methods have been added to BRepAlgoAPI\_BooleanOperation class:

- Modified2()
- Generated()
- HasModified()
- HasGenerated()
- HasDeleted()

To support the history mechanism new methods have been added to the BOP\_Builder class:

- SetHistoryCollector()
- GetHistoryCollector()

To support the history mechanism the following new method has been added to the BOP\_Section, BOP\_ShellSolid, BOP\_SolidSolid and BOP\_WireSolid classes:

- SetHistoryCollector().



 **Modeling Data**

- **GeomLib package**

A new `GeomLib_Tool` class has been added to the `GeomLib` package. This class is intended to get 2D and 3D curves and surfaces parameters by 2D and 3D points.

 **Visualization**

- `ShiftSelect(const TColgp_Array1OfPnt2d &, ...)` method has been added to the `AIS_InteractiveContext::Select(const TColgp_Array1OfPnt2d &, ...)` one. This method is intended to provide common behavior when using polygonal selection holding the SHIFT button pressed.
- Possibility to set different properties for different sides of a surface has been added.
- Possibility to set background image for `V3d_View` has been added.
- Aspect management. Possibility to use a new graphic type such as `Display`, `Drawable`, `Callback`, `RenderingContext` has been enabled.
  - New `Drawable`, `Display`, `RenderingContext` and `GraphicCallbackProc` types have been added to the `Aspect` Package.
- Window management. Possibility to use both: the window and the parent window IDs has been added.
  - A new `xParentWindow` method has been added to the `Xw_Window` class of the `Xw` Package.
  - A new `hParentWindow` method has been added to the `WNT_Window` of the `WNT` Package.
- View management. Possibility to share the same `OpenGL` context between `Open CASCADE` and another application has been enabled.
  - A new `SetWindow` method has been added to the `V3d_View` and `Visual3d_View` classes.
  - A new `Create` constructor has been added to the `Graphic3d_GraphicDevice` class.
  - A new `Begin` method has been added to the `Graphic3d_GraphicDriver` class.
  - A new `Begin` method has been added to the `OpenGL_GraphicDriver` class.

 **Test Harness**

- The `v2dsetbg` DRAW command has been added to the `ViewerTest` package. The command allows you to load a background image to the 3D Viewer.



## Improvements

### Application Framework

- **TDF package**
  - Protection (optional) against modifications outside the transaction (modified classes: TDF\_Attribute, TDF\_Data, TDF\_Label, TDocStd\_Document)
  - Improved performance of TDF\_Label::FindChild() method through fast access to the last label in the tree of labels (modified class: TDF\_Label).
  - Protection against stack overflow in the destructor of the TDF\_Label class (modified classes: TDF\_Label and TDF\_LabelNode)
- **TDocStd package**

Improved multitransaction manager (class TDocStd\_MultiTransactionManager), namely:

  - Some mistakes have been corrected.
  - A functionality of protection against modifications outside the transaction has been added.

### Data Exchange

- **Shape Healing**

Possibility to fix gaps and merge small edges in free wires (wires not belonging to faces) has been added.
- **STEP translator**

Support of parsing and formatting of STEP AP 209 entities has been added.
- **IGES translator**

Reading of faces based on Revolution Surfaces (type 120) and Extrusion Surfaces (type164) has been improved.

### Modeling Algorithms

- The behavior of the BRepBuilderAPI\_GTransform::ModifiedShape() method has been modified to make possible the retrieval of modified subshapes.
- The pipe algorithm has been modified and that allows to process some previously non-processable cases. The BRepCheck, BRepFill, Geom and GeomFill packages have also been modified.



 **Modeling Data**

- **BRepLib package**

The unreasonable tolerance increase when `BRepLib::SameParameter()` is used has been corrected.

 **Visualization**

- Display in the hidden line removal (HLR) mode in perspective view has been improved.
- Performance of multiple selections has been improved concerning the `AIS_InteractiveContext::AddOrRemoveSelected()` method.

 **Test Harness**

- The **buildsweep** command in DRAW has been improved concerning the keys "-C" and "-R". These keys specify how a pipe should be built at its "corners".
  - Key "-M" means that the pipe is continued from its final section on the previous edge along the following edge, so the pipe is "modified" at the corner. This option is set by default.
  - Key "-C" means that the two parts of the pipe are prolonged to a necessary extent and intersect at the corner.
  - Key "-R" means that the two parts of the pipe are not prolonged at the corner but are connected by a surface of revolution.

*Note : The algorithm may fail when two BSpline surfaces intersect. Therefore such case is processed with the keys "-C" and "-R" only and the failure often happens when we add a section with a law, so the pipe automatically becomes BSpline.*

 **Rebuilding Modules**

- Rebuilding of Open CASCADE Modules has been improved by means of **make** command on Linux and Sun platforms
  - Some mistakes have been corrected.
  - The make command is now taking into account necessary define options of compiler

*Note : New correct Makefile files have been added to \$CASROOT/adm/... folders.*

## Bug Fixes



- Open CASCADE 5.0.1 incorporates **74** bug fixes. For details, refer to [Appendix](#).



## Appendix: Open CASCADE 5.0.1 Bug Fixes

- [Application Framework](#)
- [Data Exchange](#)
- [Foundation Classes](#)
- [Modeling Algorithms](#)
- [Shape Healing](#)
- [Visualization](#)

<b>Application Framework, 6 corrected bugs</b>	
ID	Summary
1395	Wrong behavior of Undo / Redo and SetUndoLimit methods in TDocStd_Document
1454	Improved performance of TDF_Label::FindChild
1487	Call to Backup() is lost in TDataStd_RealArray::Init()
1722	Wrong behavior of the method CommitCommand of class TDocStd_MultiTransactionManager
1726	TDF_LabelNode::~~TDF_LabelNode causes stack overflow
2330	Protection from document changing outside the transaction has been implemented
<b>Data Exchange, 8 corrected bugs</b>	
ID	Summary
660	When reading IGES, surfaces types change causing problems with 2d mapping
663	Wrong translation of faces on SurfaceOfRevolution
916	A bug in BRepAlgo_Sewing
1773	Direction of normal on contour depends on types of curves composing the contour
1775	Protection from exception rising
1777	Improvements for supporting the STEP schema.
1779	Loading STEP AP209 entities
2102	Modification for reading from and writing to the STEP format implemented for TRJ10 TRJ11
<b>Foundation Classes, 8 corrected bugs</b>	
ID	Summary
670	An attempt to print an exception message to a stream causes crash when the message is null.
902	Wrong derivative value
1497	Removed dependence of OCC on MFC
1675	Incorrect behavior of TCollection_Array2
1723	OSD::SetSignal : not all FLT exceptions are caught
2428	A method is needed to obtain a message string from the Standard_Failure class
2499	The assignation operator for all maps is bugged
2707	Various improvements of RINA project
<b>Modeling Algorithms, 33 corrected bugs</b>	
ID	Summary
9	Bug in the pipe
358	The helical pipe is not shaded in the AISViewer
426	The result of fuse operation is invalid





433	Possible regression in BRepCheck in dev
526	BRepAlgoAPI_Section fails to build planar sections on some faces
548	Function BRepTool::IsClosed returns Standard_True although the argument is an open shell
624	Reverse Evaluator on Surface or Curve is missing
693	Boolean operations on compounds of adjoint solids fail
823	BRepAlgoAPI_Fuse fails on two cylinders
824	BRepAlgoAPI_Fuse fails on a cylinder and a sphere
826	BRepAlgoAPI_Fuse fails on a revolved shape and a sphere
827	BRepAlgoAPI_Fuse fails on a cylinder and a torus
828	BRepFilletAPI_MakeFillet fails on a prism
967	Hang up in GeomAPI_IntCS
1006	Bug in BRepAlgoAPI_Cut
1007	gp_GTrsf and function ModifiedShape of associated BRepBuilderAPI_GTransform
1226	Revolution problem reproduced in OCC Samples.
1456	Impossible to build mixed type fillets
1477	Problems in MakePipeShell
1489	Boolean operations (TKBOP) fail on given shapes
1506	Extremely large tolerance after the sameparameter operation.
1646	BRepPrimAPI_MakePrism bug
1651	No intersection found using BRepClass3d_Intersection3d
1665	Geom_BSplineSurface::SetVPeriodic throws an exception
1801	Exception is raised during the execution of command vprops in DRAW on an attached shape
1870	The shading algorithm hangs up
2044	Slow work of BRepLib::EncodeRegularity()
2056	Modification Adaptor3d_IsoCurve
2083	BRepOffsetAPI_MakePipeShell throws an exception
2095	BRepClass3d_SolidClassifier defines incorrect status of infinite point on solid.
2172	Fuse operation produces an invalid shape
2351	Problem of New BOP
2502	Bug in IntCurve_IntPolyPolyGen::Perform
2737	Redesign (improvement) of Boolean Operations History
<b>Shape Healing, 4 corrected bugs</b>	
<b>ID</b>	<b>Summary</b>
398	ShapeHealing 1:1 modification increases the number of unique shapes in a compound structure.
1430	ShapeFix_Wireframe does not process separate wires.
1133	New tool for removing locations from a shape.
1642	ShapeFix_Wire::FixSelfIntersectingEdge does not remove a loop.
<b>Visualization, 15 corrected bugs</b>	
<b>ID</b>	<b>Summary</b>
137	A patch has been integrated into Open Cascade
177	The method NotUseDisplayedObjects() from AIS_InteractiveContext class is not implemented.
230	Numeric Error occurs in V2d_View::WindowFit() Magnify()
280	Problem with view
575	V3d_View Printing

672	Wrong visualization of 2d dimensions
749	An improvement in visualization needed
1174	Impossible to set different properties to different sides of a (sur)face
1188	Impossible to set background image for V3d_View
1576	Dynamic Highlight bug
1786	AIS_LC::AddOrRemoveSelected(Handle(SelectMgr_EntityOwner) ..) is too slow
2052	AIS_InteractiveContext::Select( <polyline> ) method has no ShiftSelect() pair
2066	CRASH in AIS_InteractiveContext::PurgeViewer(const Handle(V3d_Viewer)& Vwr)
2630	Problems in OpenGL_PrimitiveArray.c
2701	OJYG4LK7 : allow any GL context to be used with Open CASCADE

