

Open CASCADE 5.1

Minor release



Release Notes

Overview

Open CASCADE 5.1 is a minor release, which includes new features, improvements and bug fixes, over major release 5.0.

Version 5.1 is binary incompatible with Version 5.0, so applications linked against the latter must be recompiled to run with 5.0. At the same time, source compatibility is almost fully retained (see [Changes](#)). Due to this binary incompatibility all applications linked against 5.0 must be recompiled to run with 5.1.

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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()`.



- **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. Please, consult the Reference Documentation for a more advanced description.

 **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)
- Memory allocation mechanism used by the TDF_Data class has been improved. The benefits of this new allocation are most noticeable on performance and memory consumption when dealing with large OCAF documents. Refer to Open CASCADE Reference Documentation for more advanced description.

- **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.

- **TNaming package**

The TNaming_Tool::CurrentShape() method and the TNaming_NewShapeIterator class (which are parts of the topological naming algorithm) have been improved to correctly process shape modification history containing deletion operations.

(Initial contribution by Sangsu Lee from MidasIT)

- A description of TDF_Data class has been added to the Application Framework section of the Reference documentation.

- **Support of Open / Save operations return status** has been added:

In the **PCDM** package:

- PCDM_ReaderStatus enumeration has been added,
- method GetStatus() retrieving a status has been added to the PCDM_Reader class,
- a new reading procedure (in PCDM_RetrievalDriver) to correctly process the status of the reading process;

In the **CDF** package:

- CDF_RetrieveableStatus enumeration has been extended with about 10 new values,



- method `GetRetrieveStatus()` retrieving a status has been added to `CDF_Application`,

In the **TDocStd** package:

- correct processing of exceptions during Open / Save operations has been added,
- the real status of Open / Save operation is now always returned,
- if an error during an attempt to open the document is fatal its `Handle` is nullified.

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.

- **Performance gain when using surface and curve adaptors**

Calculation of continuity intervals is now cached. This allows to reuse the results of previous calculations in further calls. This improvement concerns `Intervals()` and `NbIntervals()` methods of surface and curve adaptors (classes `Adaptor3d_Surface`, `Adaptor3d_Curve`, etc).

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.
- **BRepOffsetAPI package**

The history mechanism in the `BRepOffsetAPI_MakePipeShell::Generated()` method has been improved to return generated shapes more accurately. For details refer to Open CASCADE Reference Documentation on `BRepOffsetAPI_MakePipeShell`.



 **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.

- **Performance gain in graphical presentations with text objects**

Text objects can now be disabled in graphical presentations in animation mode. This allows to significantly increase the performance especially on larger presentations (when complex objects are viewed).

 **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.

- Rebuilding of Open CASCADE Modules has been improved by means of MS Visual Studio project files on Windows platform.



- Some mistakes have been corrected.
- Project dependencies within a workspace are now correctly defined (for most convenient rebuilding a project named "All" should be used).
- Disabled incremental building.

Note : See also "Changes" below.

Changes

- Concerning the API level this minor release is not fully source-compatible with its predecessor (OpenCASCADE 5.0). In particular, the name of the function in the `Standard_Failure` class returning the message string has been changed. The old name `GetMessage()` has been replaced with `GetMessageString()` in order to eliminate the conflict with a macro definition that occurs in MS Windows.
- **Rebuilding Modules**
Libraries (`.dll`, `.dld`, `.lib` and `.libd`) and executable (`.exe`) files built by MS Visual Studio projects on Windows platform, are now placed to the `ros/win32` folder and replace their corresponding original libraries and executable files.
- Beginning from this release 5.1 Open CASCADE will no longer be supported on the following platforms:
 - Windows 98
 - Linux Red Hat 6.x

Bug Fixes



- Open CASCADE 5.1 incorporates **130** bug fixes. For details, refer to [Appendix](#).



Appendix: Open CASCADE 5.1 Bug Fixes

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

Application Framework, 13 corrected bugs	
ID	Summary
1023	Impossible to run DFBrowse command in DRAW
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
2793	BinOcaf: low performance when saving documents with large attributes
2852	Impossible to determine that a document file is garbage in the current persistence
2932	Create Attribute Delta depending on the actual change of the attribute
3144	Improved performance on the allocation of TDF_LabelNode
Data Exchange, 15 corrected bugs	
ID	Summary
51	Exception when reading an IGES file after having read a STEP file
131	Problems when reading IGES
396	Bad result after applying ShapeFix_Shape on attached shape
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
2368	Invalid result after reading a STEP file
2386	Improvement of NbIntervals and Intervals of Adaptors
2941	Regression on reading a file with external references
2969	Error while reading lexically disordered entities in a STEP file
Foundation Classes, 9 corrected bugs	
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 one of the customers' project
2978	Warning/Error message appears when compiling instantiations of NCollection_Vector
Modeling Algorithms, 55 corrected bugs	
ID	Summary
9	Bug in the pipe
36	Sewing takes too long
239	Incorrect result of topological operations "bopcommon" and "boptuc" with attached shapes.
240	Topological operations "bopcommon" and "boptuc" with attached shapes returns an empty result.
241	Topological operations "bopcommon" and "boptuc" with attached shapes returns an empty result.
242	Topological operations "bopcommon" and "boptuc" with attached shapes returns an empty result.
243	Topological operations "bopcommon" and "boptuc" with attached shapes returns an incorrect result.
244	Wrong amount of vertexes in the resulting shape (13 instead of 12) after using the "bopcommon" topological operations with attached shapes.
245	Incorrect result of topological operations "bopcommon" and "boptuc" with attached shapes.
246	Wrong amount of vertexes in the resulting shape after using the "bopcommon" topological operation with attached shapes.
247	Function "bop" with attached shapes calls exception.
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
474	Impossible to build a fillet
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
615	Wrong result of BLEND operation
616	Function MKOFFSET gives an exception for an attached face
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
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
2085	Slow work of DRAW command "project" in dev
2095	BRepClass3d_SolidClassifier defines incorrect status of infinite point on solid.
2172	Fuse operation produces an invalid shape
2351	Problem of New BOP
2442	Error in BRepClass3d_SolidClassifier
2500	BRepAlgoAPI_Section has a problem with calculation
2502	Bug in IntCurve_IntPolyPolyGen::Perform
2503	Method IntCurvesFace_ShapeIntersector::IsDone(void) missing
2569	If it is impossible to create a bezier curve, it throws an exception.
2737	Redesign (improvement) of Boolean Operations History
2784	"Common" and "cut" operations for two faces give an invalid result
2785	"Cut" and "fuse" operations for two faces that share the same domain give an invalid result
2986	Cut and fuse operations. for 2 toroidal faces, sharing the same domain give an invalid result
2991	BRepMesh_IncrementalMesh takes forever to mesh some faces
Modeling Data, 3 corrected bugs	
ID	Summary
354	Neither chamfer nor blend is built in the similar situation
623	Incorrect data in PCurve
2974	Bug in parametric definition of conical surface
Shape Healing, 5 corrected bugs	
ID	Summary
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
2822	Incorrect work of FixClosed in ShapeFix_Wire
Visualization, 23 corrected bugs	
ID	Summary
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
627	2D view keeps many copies of the same object after Display()-Erase()-Display() combinations
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
1629	Viewer is always updated on closing local contexts
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
2862	V3d_View::Dump produces an incorrect result.
3021	In the animation mode, it is impossible to suppress the text display
3022	Setting the degeneration ratio to 1.0 has no impact on primitive arrays
3192	Access violation in OpenGL library
3298	Some viewer functions are not available on Windows NT
WOK, 7 corrected bugs	
ID	Summary
576	Missing automake files
577	Automake building fails
666	XSDRAWEXE configuration
667	libtool max_cmd_len
781	Compil on Solaris 2.6 with CC4.2
2994	License problem
3302	Undefined alloc symbol with compilation by *.comp scripts