

Open CASCADE 5.2.2 Maintenance Release

Release Notes

Overview

Open CASCADE Technology 5.2.2 is a maintenance release, which includes improvements and bug fixes, over previous maintenance release 5.2.1.

Version 5.2.2 is binary incompatible with any of the previous versions of Open CASCADE Technology, so applications linked against a previous version (major, minor or maintenance) must be recompiled to run with this Version 5.2.2.

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Highlights

- Lite OCAF has been implemented in order to improve performance of applications that use the OCAF component;
- Open CASCADE Technology module(s) and the whole OCCT can now be built with the `nmake` command on the Windows platform;
- The API of ThruSection algorithm has been improved.

New Features



Foundation Classes

- A new algorithm providing a binary tree of bounding boxes (`NCollection_UBTree`) has been implemented in the form of C++ template. This algorithm makes it possible to significantly increase performance of operations where the filtering of geometric objects by bounding boxes is used. It allows to reduce the computation time to a function of $N \cdot \log(N)$ when comparing all objects with each other in a set of N objects, as opposed to a direct method that takes $N \cdot N$ operations.



Modeling Algorithms

- A new tool for checking arguments of Boolean operations has been implemented (`BOP_ArgumentAnalyzer`). This tool allows to discover if a Boolean operation can/cannot be performed with given arguments or whether a Boolean operations algorithm cannot succeed due to its limitations - it assumes that the given shapes are topologically valid (by means of `checkshape` tool). This tool possesses certain options to manage parameters of checking and output format.
- The API of `ThruSection` algorithm has been improved. Creation of lofts and swept shapes with punctual sections at extremities is now available.



Application Framework

- **Lite OCAF component has been implemented**
The OCAF component uses a lot of memory during work. Even a simple application, which uses OCAF as a tool for data storing, occupies a lot of memory and reduces performance. The solution of this problem is to create new simplified (Lite or basic) OCAF toolkits.

Thus, new OCAF toolkits based on old packages, without presentation-dedicated packages have been created. Presentation-dedicated packages, such as `TPrsStd`, `PPrsStd` etc, use visualization packages that use a lot of memory. This modification will significantly decrease memory use by applications that don't need visualization.

See also: [Appendix 2](#)





Draw Test Harness

- New **bopargcheck** command has been added to DRAW Test Harness. The command allows to check the correctness of arguments for Boolean Operations.

Use `>bopargcheck Shape1 [[Shape2] [-F/O/C/T/S/U] [/R|F|T|V|E|I|P]] [#BF]`

-<Boolean Operation>

- F (fuse)
- O (common)
- C (cut)
- T (cut21)
- S (section)
- U (unknown)

For example: "bopargcheck s1 s2 -F" enables checking for Fuse operation. Default - section.

/<Test options>

- R (disable small edges (shrank range) test)
- F (disable faces verification test)
- T (disable tangent faces searching test)
- V (disable test possibility to merge vertices)
- E (disable test possibility to merge edges)
- I (disable self-interference test)
- P (disable shape type test)

For example: "bopargcheck s1 s2 /RI" disables small edge detection and self-intersection detection. Default - all options are enabled.

#<Additional Test Options>

- B (stop test on first faulty found); default - OFF
- F (full output for faulty shapes); default - output in a short format

NOTE: <Boolean Operation> and <Test Options> are used only for a pair of argument shapes, except I and P options that are always used for a pair of shapes as well as for single shape test.

Improvements

Foundation Classes

- Some boolean operations memory leaks have been fixed.

Modeling Algorithms

- Treatment of 3d offsets on "lofts" (shapes constructed by the Thrusections algorithm) has been improved.

Visualization

- Selection mechanism has been improved in order to clear the current selection after selecting an empty space with a rectangle when the local context is open.
- Visualization algorithm has been modified in order to allow simultaneous display of Bitmap fonts, Texture Mapped fonts and Textures.

Products

- Several improvements including performance improvements have been implemented and additional algorithms have been added to the Open CASCADE Mesh Framework.
- The QMchecktopo DRAW command has been corrected in order to output information messages only if errors are detected.

Samples

- A possibility to save and open files in binary and xml formats has been added to the MFC OCAF sample.

Building Tools

- A possibility to build OCCT modules and the whole OCCT with the nmake command on Windows platform has been implemented.

WOK

- WOK has been corrected in order to fix the problem with the xcpp.template wok compilation step on Windows platform.

Changes

✓ CAD Models

- Starting from OCCT 5.2.2 CAD models (.brep, .iges, etc) are distributed within a special \$CASROOT/./data folder of Open CASCADE Technology distribution package. All "samples/data" folders have been eliminated from the package.

✓ Visualization

- Currently Texture Mapped fonts are used by default. In order to switch the default font type it is possible to use `TexFontEnable` or `TexFontDisable` static methods from the `Graphic3d_AspectText3d` class. To switch texture mapped fonts on/off the `Graphic3d_AspectText3d::SetTextureMappedFont (Standard_Boolean)` method should be used with the `Standard_True/Standard_False` argument. First, a `Graphic3d_AspectText3d` object should be created and initialized and then this object can be used in a usual manner, i.e. passed to the `Graphic3d_Group::SetPrimitivesAspect()` method. Without explicit calls to `Graphic3d_AspectText3d::SetTexturedMappedFont()` the objects get created with the use of texture mapped fonts (if `TexFontDisable` was not called previously).

✓ Application Framework

- Lite OCAF has been implemented involving the following changes to Open CASCADE Technology components:
 - TDocStd, TDataStd, TDF, TNaming, TFunction packages moved to TKLCAF toolkit;
 - PDF, PDocStd, PNaming, PDataStd, MDF, MDocStd, MDataStd, MNaming packages - moved to TKPLCAF toolkit;
 - StdSchema - several packages removed;
 - PCDMShape package - moved to TKShapeSchema toolkit;
 - PAppStd package - renamed into StdDrivers;
 - PAppStdPlugin executable - renamed into StdPlugin;
 - BinDrivers - several classes removed;
 - BinMDF, BinMDataStd, BinMNaming, BinObjMgt, BinMDocStd, BinTools packages - moved to toolkit TKLBin;
 - XmlDrivers - several classes removed;
 - XmlMDF, XmlMDataStd, XmlMNaming, XmlObjMgt, XmlMDocStd packages - moved to toolkit TKLXml.

See also: [Appendix 2](#)

✓ Data Exchange

- A modification has been made to avoid setting the DONE status for a valid shape after `ShapeFix_Shape`.

✓ Building Tools

- TKjcas toolkit has been moved from Foundation Classes module to a separate module Wrappers.





Porting

- Contribution from Anton Verburg, Open CASCADE Community. Open CASCADE Technology Compilation has been provided on Debian 1:3.3.4-13 with gcc 3.3.4 compiler.

This configuration is not currently planned to be officially supported in the future releases. It is only a signal that version 5.2.2 works on that configuration with the same quality as on any other officially supported configurations.

Bug Fixes



- Open CASCADE 5.2.2 incorporates **39** modifications (bug fixes, enhancements and other corrections) over version 5.2.1. For details, refer to [Appendix 1](#).



Appendix 1: Open CASCADE 5.2.2 Bug Fixes

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

- [OMF](#)
- [Express Mesh](#)
- [Collision Detection](#)

Foundation Classes, 4 bug fixes	
ID	Short Description
7034	NCollection List and Sequence modified in SD project
7253	Compilation at g++ (GCC) 3.3.4 (Debian 1:3.3.4-13)
7287	Problem of Memory Leak
7349	New algorithm for binary tree of bounding boxes
Modeling Algorithms, 11 bug fixes	
ID	Short Description
593	Projection of a 3D point on surface using GeomAPI_ProjectPointOnSurf works incorrectly
5157	DRAW commands vprops and srops with tolerance 1.e-6 huge on attached shape.
5806	BRepOffsetAPI_MakeThickSolid fails to build a result
6538	Empty result obtained by operation "common" for couple of faces
6554	Intersection shell-plane gives a disconnected set of edges
6811	Unstable work of the checkshape on the attached file.
6841	Invalid shape produced by Boolean common
6842	Stack overflow in Gluing operation
6862	Tool for checking arguments of Boolean Operations
6891	Triangle algorithm exits the application
7166	Improvement of API of ThruSection
Visualization, 7 bug fixes	
ID	Short Description
5988	A textured shape (AIS_TextureShape) becomes black if we display an AIS_PlaneTrihedron with it.
6571	Bitmap and Texture mapped fonts optimization
6802	Exceptions in visualisation when MMGT_OPT=0 and MMGT_CLEAR=0



7089	Sometimes the text is not displayed when using texture mapped fonts.
7186	Selecting an empty space with rectangle doesn't clear the current selection, when there is an opened local context
7190	Texture-mapped fonts don't work, as soon as any old-style (bitmap) text is displayed
7288	Texture with text
Application Framework, 1 bug fix	
ID	Short Description
7262	Implementation of Lite OCAF
Shape Healing, 4 bug fixes	
ID	Short Description
5624	Exception during usage DT_ShapeConvertRev command
6366	FixShape does not fix an edge with invalid SameRange flag
6555	ShapeFix_Shape modifies valid shape and returns wrong status DONE.
6810	Duplication of code in ShapeFix_IntersectionTool
Data Exchange, 2 bug fixes	
ID	Short Description
6497	Bug in IGESControl_Reader::PrintTransferInfo
7141	STEP interface crashes exporting large assembly
Test Harness - Draw, 1 bug fix	
ID	Short Description
7021	Two different draw-commands have the same name
WOK, 2 bug fixes	
ID	Short Description
6789	WOK can not generate projects and solutions for MS VC7.1
7286	WOK step xcpp.template does not work on WNT
Building Tools, 1 bug fix	
ID	Short Description
7302	Creation of possibility to generate *.mak files in order to build whole OpenCascade with MS nmake. Moving of TKjcas to a separate module Wrappers



Product Bug Fixes

The following bug fixes have been performed for Open CASCADE specific development customers.

Open CASCADE Mesh Framework, 4 bug fixes	
ID	Short Description
880	Incorrect work of SMDS_MeshNode::RemoveInverseElement function
7022	Updates of OMF libraries due to current development for ***
7243	AddFaceWithID works wrong if an already existing face is added with a new ID
7488	Tetrahedron creation exception
Express Mesh, 1 bug fix	
ID	Short Description
6800	Free edges are left after meshing of a sewed shell
Collision Detection, 1 bug fix	
ID	Short Description
6484	Existing collision hasn't been found by ColProxy component

Appendix 2: Lite OCAF – current changes

Transient OCAF packages and Standard persistence.

A new package AppStdL has been created, which contains one class: AppStdL_Application. This class inherits TDocStd_Application class and implements methods Formats() and ResourceName(). This class supports OCC-StdLite, BinLOcaf and XmlLOcaf formats and uses StandardLite resource file.

Toolkit TKLCAF has been created, which includes packages AppStdL, TDocStd, TDataStd, TDF, TNaming, TFunction that have been removed from TKCAF toolkit.

Toolkit TKPLCAF has been created, which includes packages of TKPCAF except PPrsStd, MPrsStd which have been removed from TKPCAF.

New StdLSchema schema has been created, which includes packages:

- PDF
- PDataStd
- PNaming
- PDocStd
- PCDM

All packages have been removed from StdSchema except PPrsStd PFunction.

New package StdLDrivers has been created, which contains the same classes as PAppStd does, but any call of MPrsStd and MFunction methods is excluded.

Methods Factory() of PAppStd and StdLDrivers have been modified to use the nested schema mechanism.



PAppStd now uses ShapeSchema, StdLSchema and StdSchema. StdLDivers now uses ShapeSchema and StdLSchema.

Package PCDMShape has been removed from TKPCAF toolkit and added to the TKShapeSchema toolkit.

New toolkit TKStdLSchema has been created, which includes the StdLDivers package and StdLSchema.

New executable StdLPlugin has been created, similar to PAppStdLPlugin, but the former uses new packages, schema etc.

Package PAppStd has been moved to StdDrivers, executable PAppStdPlugin has been renamed to StdPlugin.

Package PCDMShape has been moved from TKPCAF to TKShapeSchema toolkit.

New resource file StandardLite has been created, and the existing plugin file has been adapted to the new format.

Binary and Xml persistence.

Binary Lite persistence.

New package BinLDivers has been created, which contains same classes as BinDrivers.

In BinLDivers.cxx file, in function BinLDivers::AttributeDrivers() calls of BinMFunction and BinMPrsStd have been removed. New GUID for Storage and Retrieval drivers has been defined in the BinLDivers.cxx file. These GUIDs have been used in the StandardLite and Plugin files.

New format (formatname: **BinLOcaf**, extension: **cbfl**) has been described in the StandardLite resource file.

All classes have been removed from the BinDrivers package.

In package BinDrivers two classes have been created: BinDrivers_DocumentRetrievalDriver and BinDrivers_DocumentStorageDriver, which inherit from BinLDivers_DocumentRetrievalDriver and BinLDivers_DocumentStorageDriver classes correspondingly. These classes (from package BinDrivers) redefine virtual methods AttributeDrivers in each class. Calls of BinDrivers::AttributeDrivers() in AttributeDrivers method of each class have been added.

New toolkit TKBinL has been created, which contains package BinLDivers and the same packages as TKBin does, except BinDrivers, BinMFunction, BinMPrsStd packages.

Packages BinMDF, BinMDataStd, BinMNaming, BinObjMgt, BinMDocStd, BinTools have been removed from toolkit TKBin.

New executable BinLPlugin has been created, similar to BinPlugin, but uses BinLDivers package.

Xml Lite persistence.

Xml persistence has been divided in two parts using same rules as are used for Binary persistence.

FormatName: **XmlLOcaf**, extension: **xmll**

