

Open CASCADE 5.2.4 Maintenance Release

Release Notes

Overview

Open CASCADE Technology 5.2.4 is a maintenance release, which includes improvements and bug fixes, over previous maintenance release 5.2.3.

Attention! Version 5.2.4 is:

- **structurally** incompatible with any previous version of Open CASCADE Technology, so applications linked against a previous version (major, minor or maintenance) must be ported to this version 5.2.4 in order to become compatible with it. See [Appendix 2](#) for detailed information.
- **binary** incompatible with the previous versions of Open CASCADE Technology, so applications linked against a previous version must be recompiled to run with this Version 5.2.4.

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Highlights

- Open CASCADE Technology has been repackaged;
- Open CASCADE Technology is now certified for the gcc 3.4 compiler on Mandrake 10 platform;
- XDE persistence classes have been upgraded to completely support the Std, Xml and Binary persistence;
- OMF (Open CASCADE Mesh Framework) product has been improved in mesh operations, visualisation and data exchange with NASTRAN

New Features



Data Exchange

- ❑ A new binary format for storing XDE documents has been implemented in Open CASCADE Technology. Its name is BinXCAF and the extension is xbf.

There are also new storing/retrieving drivers for the following XDE attributes in XML format:

- XCAFDoc_ColorTool Driver
- XCAFDoc_DocumentTool Driver
- XCAFDoc_LayerTool Driver
- XCAFDoc_ShapeTool Driver



Test Harness

- ❑ New DRAW command DFBrowser has been added to OCAF commands.

OCAF Browser is a separate tool and can be accessed from Open CASCADE through the DebugBrowser. hxx include file and DFBrowser library.

If this tool is installed during OCCT installation, it will be located in the \tools\ocafbrowsers folder. The command is added only if the PATH environment variable on Windows platform (or LD_LIBRARY_PATH on SUN or Linux) contains a path to DFBrowser.dll (libDFBrowser.so on SUN or Linux). This variable should also contain a path to FLTK or QT graphic library, and the choice of library to use depends on the version of OCAF Browser (FLTK or QT accordingly).

If DFBrowser.dll (libDFBrowser.so) or any dependant library cannot be found, a message *"Cannot load Debug Browser library. DFBrowser command is not attached"* appears in the console.

Example of launching the OCAF Browser:

```
NewDocument Doc MDTV-Standard
```

```
...
DFBrowser Doc
```



- Automatic maintenance of inverse connections has been introduced in `SMDS_Mesh`. Now the inverse connections are automatically built when a node is removed. It is not necessary to call method `'RebuildAllInverseConnections'` before executing the mesh edit operations. The data consistency is maintained automatically by `SMDS_Mesh` class. Visualization of the mesh has been improved: only the elements of the target mesh are shown; detection of free nodes has been improved. Features that allow automatic maintenance of inverse connections have been added:

In the SMDS package:

- method `RemoveAllInverseConnections()` destroying all inverse connections has been added to the `SMDS_Mesh` class;
- optional parameters `'OnlyFree'` and `'DeleteFreeNodes'` have been added to the method `RemoveNode` of the class `SMDS_Mesh`. The default value of `OnlyFree` is false. If it is true, the node will be removed only if it is free (has no inverse connections). If `OnlyFree` is false and there are elements built on this node then this elements are also removed. In addition, if parameter `DeleteFreeNodes` is true then the nodes becoming free are also removed. The default value of `DeleteFreeNodes` is true.

In the SMDSCaf package:

- optional parameters `'OnlyFree'` and `'DeleteFreeNodes'` have been added to the method `RemoveNode` of the class `SMDSCaf_TMesh`. The sense and default values of the parameters are the same as in `SMDS_Mesh`.

In the SMDSTest package:

- new command `'removeinverse'` destroying all inverse connections has been added to the `SMDSTest_ModelCommands`;
- optional Boolean parameters for removing only free node and the nodes becoming free has been added to the command `'removenode'` of the `SMDSTest_ModelCommands`;
- optional Boolean parameters for removing only free node and the nodes becoming free has been added to the command `'TMRemoveNode'` of the `SMDSTest_CafCommands`.

- The NASTRAN reader class in OMF (`SMDSTool_NASReader`) has been completely reimplemented; now it works faster, supports many (though not all) possible ways of NASTRAN data file encoding, and is able to read more entity types (including 1D elements). It can be further extended to read more information (by inheriting `SMDSTools_NASMeshReader` class).

In addition, a simple tool for writing SMDS mesh to NASTRAN has been implemented (class `SMDSTools_NASWriter`).

- Two types of mesh visualization in the shading mode have been added in OMF.

In the SMDS package:

- method `IsNormalDefined()` defining the presence of computed normals for mesh nodes of the mesh face has been added to the `SMDS_MeshElement` class



In the SMDSTools package:

- overloaded method `ComputeAreaAndNormal()` has been created for convenience. It takes `TCol gp_Array1OfXYZ` as an argument (if the mesh face is defined by array of its nodes coordinates).

This method calculates the normal and the area of the mesh face.

Improvements

✓ Visualization

- `StdPrs_Tool RFace::Next()` has been improved to avoid unnecessary recursion.
- `Graphic2d_Primitive.cdl` has been corrected for efficient compilation on Gcc3.4.

✓ Data Exchange

- The retrieving algorithm of `gp_Trsf` in Xml format has been corrected.

✓ Foundation Classes

- The template classes in the `NCollection` package have been made compilable under gcc 3.4.
- `NCollection_UBTree`: an optional parameter "theAllocator" has been added in the constructor (by default this parameter is taken as `NCollection_BaseAllocator::CommonBaseAllocator()`). This Allocator is stored as a member of `UBTree` class and can be queried by method `Allocator()`.

Similarly, method `Clear` takes one optional parameter "aNewAlloc". This is the new allocator instance that would be used from the moment of destroying the previous data allocated on the previous allocator. This feature is useful if `InlineAllocator` is employed.

`NCollection_UBTreeFiller` allocates its internal `Sequence (mySeq)` using `NCollection_InlineAllocator`, this also improves the performance.

In all three header files the comments are reformatted to the Javadoc standard (to be usable by Doxygen).

✓ OCAF

- `XmlLDivers_DocumentStorageDriver` class: ability to write the "reference section" of `XmlOcaf` documents has been added. `XmlLDivers_DocumentRetrievalDriver` class: ability to read the "reference section" of `XmlOcaf` documents has been added.
- Memory leak has been removed when generating messages during persistence storage/retrieval operations.



Modeling Algorithms

- ❑ Some memory problems (non-initialized memory reading and memory leaks) have been found and corrected in modeling algorithms. However, they did not impact the functionality of algorithms in the standard memory management mode (MMGT_CLEAR=1).



Test Harness

- ❑ A new draw command 'vpick' has been implemented for Windows platform.
Purpose: the command stores in the given arguments the coordinates of the mouse cursor in 3D Viewer. To pick the coordinates press Control+Left Mouse Button in the 3D Viewer.
Syntax: vpick X Y Z



OMF

- ❑ Two types of mesh visualization in the shading mode have been added in OMF:
 1. Default shading mode
 2. Smooth shading mode
 - 1) In this mode the mesh presentation computes normals for each face of the mesh (the normal for each mesh node of this face is the same).
 - 2) Smooth shading mode is available only if normals for each mesh node are already computed (if the mesh is not obtained from STL data but converted from a shape) otherwise the default shading mode is used.
- ❑ STL writer has been corrected so that to compute right normals to triangles before writing them to a file.





Changes

✓ General

The contents and the structure of some packages and toolkits in Open CASCADE Technology have been revised so as to reduce the number of cross-toolkit dependencies, thus reducing the amount of OCCT libraries necessary to be linked with many applications. The code has been cleared by removing or separating some obsolete, duplicated, or rarely used items.

Most important changes :

Class Sewing has been moved from package BRepAlgo (TKBool toolkit) to package BRepBulderAPI (TKTopAlgo), as a result sewing can now be used without need to link with huge libraries providing Boolean operations.

Class BRepOffsetAPI_Sewing is now alias (typedef) to BRepBulderAPI_Sewing. Please be aware that the constructor and method Init() in class BRepBulderAPI_Sewing possess one argument more than in class BRepOffsetAPI_Sewing (second argument); to reproduce the same behavior that argument should be set to True.

All classes from package MoniMessage and classes implementing the progress indicator in package MoniTool have been renamed and put into a single new package Message in TKernel. This is done to facilitate the usage of these general-purpose tools in applications. Please, see [Appendix 2](#) for detailed information.

✓ Foundation Classes

- ❑ The macros identifying the current version of Open CASCADE library (defined in Standard_Version.hxx) has been extended by an additional macro identifying maintenance version number (OCC_VERSION_MAINTENANCE). Note that macro OCC_VERSION still includes only minor and major version numbers, but not the maintenance version number.

✓ Modeling Algorithms

- ❑ The toolkits TKOffset and TKShHealing depended on TKDraw in debug mode on Unix platforms. It required presence of the shared object libtcl8.3.so to run the OCCT libraries compiled in debug mode. Now this dependence has been removed.

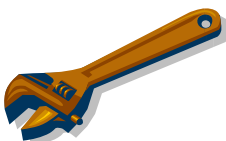
✓ Visualization

- ❑ Internal TEL_COLOUR structure declared in InterfaceGraphic_tel em. h file and widely used in Open CASCADE Technology visualization engine has been changed to include alpha value, in addition to three color components.

✓ OCAF

- ❑ Method PCDM_Referencerator::Init() has been modified - private status has been removed.

Bug Fixes



- ❑ Open CASCADE 5.2.4 incorporates **35** modifications (bug fixes, enhancements and other corrections) over version 5.2.3. For details, refer to [Appendix 1](#).





Appendix 1: Open CASCADE 5.2.4 Bug Fixes

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

Foundation Classes, 4 bug fixes	
ID	Short Description
8759	Compilation error with gcc3.4 at NCollection_Sequence.hxx compilation
9047	Addition of numbering of maintenance version into Standard_Version.hxx
9059	Problems building 5.2.3 on .NET in Debug mode
8682	General revision of OCCT packaging
Modeling Algorithms, 7 bug fixes	
ID	Short Description
8712	gcc3.4 porting
9131	Regress on Mandrake 10 cfe 900 J6
9134	Regress on Mandrake 10 cfe 903 A7
9145	Regress on Mandrake 10 chl 922 V4
9444	Memory problems (UMR and MLK) in some modeling algorithms
9498	Remove dependency of TKOffset and TKShHealing on TKDraw
7668	BRepOffsetAPI_MakeOffset wrong generated list
Visualization, 6 bug fixes	
ID	Short Description
8563	Incorrect comparison in StdPrs_ToolShadedShape::Normal
8722	Optimization of StdPrs_ToolRFace::Next() method
8757	gcc3.4 returns error at Graphic2d_Primitive.hxx compilation
8854	Regression in primitive arrays
9127	Visualization problems on Mandrake 10 platform
9489	Very low performance of MeshVS presentation on preselection
Application Framework, 4 bug fixes	
ID	Short Description
7724	Compilation error with gcc3.4 at PCDM_RetrievalDriver.hxx
8758	gcc3.4 returns error at Graphic2d_Primitive.hxx compilation
9060	OCAF is crashed after NewCommant DRAW command on Linux mandrake 10
9346	Memory leak in CDM_COutMessageDriver::Write



Data Exchange, 4 bug fixes	
ID	Short Description
8229	Exception is raised during the ReadStep in DRAW on the attached shape
9123	Problems on dec (XDE) grid tests on Linux Mandrake 10
9141	Regress on Mandrake 10 chl 921 A6
9270	Upgrade of XDE persistence
Shape Healing, 2 bug fixes	
ID	Short Description
9129	Regress on Mandrake 10
9274	REGRESSION of ShapeAnalysis_FreeBounds on Mandrake
Test Harness, 2 bug fixes	
ID	Short Description
8662	Implementation of draw command vpick.
9373	Convenient tool for navigating OCAF document under Draw
OMF, 6 bug fixes	
ID	Short Description
7664	Normals are calculated incorrectly during importing of mesh as STL file
8278	Exception after removing a node in a mesh with children
8527	Visualization error when a patch neighboring to a free edge is deleted
9115	Introduce smooth shading mode
9535	Incorrect visualisation of volumic mesh using SMDSInter_Mesh
9583	Improve OMF by improving NASTRAN reader



Appendix 2: General Revision of OCCT Packaging

Complete list of changes

Removals

The following obsolete classes and units have been removed from OCCT. If some of them are used in an application, the suggestion for replacement is given in square brackets:

- toolkit TKMoni Frame [most useful part of its functionality is available in ShapeProcess package]
- packages Correct, CorrectTest, CheckWire [the relevant functionality is now provided by the ShapeHealing toolkit -- see classes ShapeAnalysis_Wire, ShapeFix_Wire and others]
- package MgtBRepAbs [the enumeration TriAngleMode moved to package MgtBRep]
- class BRepTools_Sewing [class BRepBuilderAPI_Sewing should be used instead]
- package ShapeInterference [the relevant functionality is covered by Collision Detection product]
- package ShapePlacement
- classes Prs3d_WFSectionShape, StdPrs_ToolSectionShape and StdPrs_WFSectionShape

Note that in case of strong necessity to use some of these items their sources can be taken from a previous version of Open CASCADE Technology.

Logical movements

The toolkit TKShHealing has been moved from Data Exchange module to Modeling Algorithms.

The following packages have been moved to toolkits where they are more appropriate. Note that these changes do not affect compilation; they can lead only to a necessity to modify a set of libraries the application is to be linked to.

- Package LibCl: TKernel -> TKXSBase
- Packages Contap, TopBas and TopCnx: TKTopAlgo -> TKHLR
- Package TColQuantity: TKernel -> TKService
- Packages Prs2d, GGraphical2d, AIS2D, and Select2D: TKV3d -> TKV2d
- Package BRepProj: TKOffset -> TKBool
- Package MoniTool: TKShHealing -> TKXSBase
- Package TopLoc: TKBRep -> TKMath
- Package BRepAlgoAPI: TKBool -> TKBO

Renaming

The following packages and classes have been moved to other toolkits (packages) and renamed. Any code that makes use of relevant classes needs to be corrected.

- Package MoniMessage (TKShHealing) -> Message (TKernel)
- Package TopOpBRepApprox (TKBool) -> BRepApprox (TKTopAlgo)

- Class Cl ass2d: BRepTopAdaptor -> CSLi b_Cl ass2d
- Class ProgressI ndicator and related classes: Moni Tool (TKShHeal i ng) -> Message (TKernel)
- Class Posi ti on: *PrsStd -> *DataStd, for all packages with * replaced by T, P, M, Xml M, Bi nM

Separation

New toolkit TKAdvTools has been created to contain some specific packages moved from TKernel (Dynami c, Materi al s, Graph, GraphDS) and TKMath (Expr, ExprI ntrp)

New toolkit TKMesh has been created to collect packages related to support of mesh data structures and algorithms: packages MeshDS from TKBRep toolkit; Tri angl e, MeshAl go, MeshShape, BRepMesh, and I ntPol y from TKTopAl go toolkit.