



Open CASCADE Technology and Products 6.4

Release Notes

Overview

Open CASCADE Technology and Products version 6.4 is a minor release, which includes new features, improvements and bug fixes, over minor release 6.3 and maintenance release 6.3.1.

Version **6.4** is binary incompatible with the previous versions of Open CASCADE Technology and Products, so applications linked against a previous version must be recompiled to run with this Version 6.4.





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Highlights

- **Open CASCADE Technology**
 - Accelerated triangulation algorithms in BRepMesh package, due to using Intel® TBB external library for the paralleling tools and the memory manager.
 - Use of freetype and ftgl external libraries for advanced 2D and 3D text visualization.
 - Improvement of algorithms for line-line, line-plane and plane-plane intersection.
 - New quaternion class for definition and manipulation of 3D rotation operators.
 - New interactive object class in AIS package for displaying triangulation encapsulated in Poly_Triangulation object.
- **Products**
 - New BestFit product, which computes the transformation minimizing the normal distance between the given cloud of 3d points and the given shape.
 - New types of Boolean operations for surface meshes in Open CASCADE Mesh Framework (OMF) product.
 - New methods to read and write SAT data to streams in Open CASCADE ACIS SAT Interface product.



Foundation Classes

- The new algorithmic class `Poly_MakeLoops` allows finding all minimal loops in a graph. The link in the graph is defined as a pair of node indices. The user defines a special callback class that provides the algorithm with all necessary geometric definitions. There are two descendants of the algorithm, one for working in the 2D space, and another for working in the 3D space on a surface, normal directions to which are defined at nodes.
- The new class was derived from `NCollection_BaseAllocator`. It uses the global dynamic heap (`malloc / free`). It will allow implementing algorithms that use the most suitable for the caller method of allocation (OCCT standard allocator, `IncAllocator` or `direct malloc`). The name of the new class is `NCollection_HeapAllocator`.
- OCCT memory manager has been extended by the possibility to use Intel® TBB memory allocator (if TBB library is available). For this, environment variable `MMGT_OPT` should be set to 2. It is recommended to use this option in applications that may call OCCT functions from multiple threads. Note that it is also necessary to set environment variable `MMGT_REENTRANT` to 1.
- A new class of quaternions has been introduced into OCCT. Quaternions are convenient as a fast and robust approach to definition and manipulation of 3d rotation operators. The new `Quaternion` class has been added in `gp` package and is used in `gp_Trsf` class to provide more precise and convenient rotation operations.
- New static methods `Color2argb` and `Argb2color` have been added to the class `Quantity_Color`. They convert an object of type `Quantity_Color` to the integer ARGB value and vice versa. ARGB is an abbreviation for Alpha (transparency coefficient), Red, Green, Blue. Each property occupies 8 bits in the integer result. In conversions to and from `Quantity_Color` class, the Alpha value of ARGB integer is ignored.
- The global function `ACosApprox` has been introduced for quick calculation of the approximate value of the arc cosine of a real argument.
- The new method `Poly::Catenate()`, which allows merging a list of `Poly_Triangulation` objects into one `Poly_Triangulation`, has been added.
- New `OSD_Localizer` class, which allows changing the system locale, has been implemented. This class can be used during export of 3D scene contents to a vector file format to ensure that decimal separators are exported correctly.
- `NCollection_Vector` has been modified to use an external allocator (passed through an optional argument in the constructor) instead of `malloc/free`. This allows tuning the performance of the applications that use the vector container.
- The new method `OSD_Thread::Wait()` allows the user to define the time for waiting before obtaining the result of the thread.
- The method `Standard::Purge` has been improved. The previous implementation of this method in the class `Standard_MMgrOpt` released only free memory blocks of sizes greater than the value of `MMGT_CELL_SIZE` (200 by default). Smaller blocks were allocated to so called memory pools, and they were never returned to the system. With this improvement, the method `Purge()` finds the pools that do not contain any busy blocks and releases them. After that the virtual address space is really returned to the system.
- The following bug in `OSD_Path` has been fixed: when initialized by a string containing three or more entries (file or directory names), with something other than `'..'` as the last entry, and `'..'` as the last-by-one entry, that last-by-one entry was lost from the path.

Note that because of this bug OCAF was unable to load a document from the file with a relative path like ". . \. . \Data\test.cbf".

- The problem causing the loss of path separator in the class `OSD_Path` has been fixed. The separator was lost by method `SystemName()` in the paths consisting of a one-letter directory and a single file (or subdirectory) name. For instance, `"/File.ext"` was misinterpreted as `".File.ext"`.
- The problem in copy function (operator `=`) of all map data structures in `NCollection` package has been fixed: the target map was resized using the number of buckets of the source map instead of its actual content (extent). This led to unnecessary growth of allocated memory whenever the map was copied.
- The implementation of classes `OSD_Chronometer` and `OSD_Timer` has been revised:
 - Two static methods `GetThreadCPU()` and `GetProcessCPU()` have been added in class `OSD_Chronometer`, allowing to query CPU times (both user-mode and kernel-mode) for the current thread or process, respectively (as real values, in seconds). Note that practical precision of this quantity for process is around 10 ms on all tested systems (Windows Vista/MSVC, SUN Solaris 10, Linux/GCC), while granularity for thread times is much less on Linux and SUN (~1 mks).
 - A Boolean parameter has been added to constructor of `OSD_Chronometer` class; when it is `True`, it will measure CPU times of the current thread instead of the process (default). `OSD_Timer` always measures the whole process CPU.
 - Elapsed time measurement precision in `OSD_Timer` has been increased (practical granularity is ~1-10 mks)
 - The code has been simplified and harmonized between platforms.
- The problem with method `gp_Trsf::GetRotation()` has been fixed: when the matrix contained a zero rotation component, this method failed (returned `False`). Now it always returns `True` and correctly sets the output vector (as unitary) and angle values.
- `NCollection_IncAllocator` has been modified to raise exception if no memory available. This allows processing low memory condition in the upper (application) level code. Earlier the exception was raised only in debug mode.
- Several minor improvements have been made in `NCollection` package:
 - `NCollection_Handle` class: avoid ambiguity of constructors when initialized by 0; provide cast operator to pointer to contained type
 - `NCollection_UBTreeFiller`: avoid compiler warning on impossibility to create assignment operator automatically (MS Visual C++ 9.0)
 - `NCollection_BaseCollection`: virtual destructor provided to ensure correct destruction for descendants
- The macro `GetSString` has been removed from `Standard_SStream.hxx`. `Draw_Interpreter` class has been extended by operator `<<` accepting stream, the text from exception object is retrieved using `GetMessageString()` method of `Standard_Failure` class. Additionally, some methods have been completely rewritten.
- All fields of type `double` in classes of `gp` package are now initialized with zero values to avoid runtime problems.
- Truncate operation in `InitFontDataBase()` method from `OSD_FontMgr.cxx` erroneously cut the full font name because of a hardcoded number of symbols. Now the result of `SearchFromEnd()` is stored in a variable and used in truncation.

- A few minor improvements have been made in the headers of Standard package:
 - Standard_Macro.hxx: automatically defines macro `_OCC64` on 64-bit AMD and Intel platforms
 - Standard_ExtCharacter.hxx: method `ToExtCharacter()` has been corrected to handle symbols from the extended part of ASCII table properly
- The interface of the class `Message_ProgressSentry` has been extended by additional arguments to methods `Next()`, allowing to change the name of the current step, and method `Show()`, which allows updating the view of the progress indicator. This improvement allows using sentry class for all manipulations with the progress indicator, which is more consistent, safe, and convenient, than direct calling its methods.

Some non-compatible changes have been made:

- Alternative method names `NextScope()` have been abandoned; it is better to use `Next()` instead
- Method `EndScope()` has been renamed to `Relieve()` to avoid confusion
- Minor corrections in various classes:
 - Several methods became virtual to allow sub-classing;
 - `gp_Quaternion` has been corrected to use a normalized value when necessary;
 - `TObj` has been protected against null model.



Modeling Data

- The checks for initial point being inside the parametric limits of the surface or curve have been removed from class `Extrema_GenLocateExt` to avoid exceptions caused by rounding errors.

Modeling Algorithms

- It has become possible to create a thick solid from a shell using the new parameter `Thickenig` in the constructor of the class `BRepOffset_MakeOffset` (and in the method `(Initialize)`).
- Tools for conversion of surfaces to B-Splines provided by `ShapeCustom` package (method and class `ConvertToBSpline`) have been extended by the possibility to convert planar surfaces. For that, an additional parameter is added (False by default). The corresponding DRAW command `DT_ToBspl` is extended by the option to specify the types of surfaces to be converted.

Implementation of method `ShapeConstruct::ConvertSurfaceToBSpline()` has been changed to use the method `GeomConvert::SurfaceToBSplineSurface()` for conversion of elementary surfaces. This improves the quality of resulting b-splines.

- A special flag permitting to take into account internal edges during the analysis of free bounds has been implemented (used in SALOME). The default behavior of `ShapeAnalysis_FreeBounds` functionality has not been changed.
- The Delaunay triangulation algorithms implemented in `BRepMesh` package have been revised and now are able to run in parallel threads on multi-core and multi-processor systems using Intel TBB library both for the paralleling tools and the memory manager.

Parallel processing is available on WNT and Linux only and requires installed TBB library. It can be done when `Standard::IsReentrant()` is True; otherwise (or when TBB is not available) a usual cycle is applied.

The availability of TBB library is optional. Dependencies from TBB library are switched on by definition of `HAVE_TBB` macro. In OCCT 6.4 package for Windows delivered with source and binary VC projects this macro is defined by default. During building on UNIX by `Makefiles` it will be necessary to define `-with-tbb-include` and `-with-tbb-library` options for configure.

- Some algorithms for line-line, line-plane and plane-plane intersection have been improved.
- The template classes that are instantiated just once and their instantiations have been replaced with usual classes. Three new classes: `Adaptor3d_HSurfaceTool`, `BRepBlend_BlendTool` and `IntPatch_SequenceOfPoints` have been created and several more classes modified
- The inconsistent behavior of class `BRepTools_Modifier` which sometimes increased tolerances of the result shapes has been improved. Now the class does not change tolerances. Please, note that this can reveal some problems with shape tolerances in the applications, which might have been previously hidden due to using this class.
- The problem with classes `BRepExtrema_DistanceSS` and `BRepExtrema_DistShapeShape`, which in some cases returned a solution with swapped parameters, if the second argument shape contained edges built on C0 curves, has been fixed.
- The problem, which caused exception in intersection algorithms (projection of edge on face) on the shapes containing degenerated edges, has been eliminated.
- The problem with `BRepTools_Modifier` causing creation of invalid shapes when two connected faces are built on the same closed surface has been fixed. The problem was that the edge between such two faces was not considered seam, and a single p-curve has been assigned to it on both faces. Now this situation is identified and processed correctly.

- Earlier the distribution direction computed for curves by classes `GCPnts_QuasiUniformDeflection` and `GCPnts_UniformDeflection` was inconsistent with the class description (the implementation depends on the curve type). Now the computed distribution always starts from the minimum and ends with the maximum curve parameter limit specified by the user regardless of the curve type (as given in the class description).
- The class `BRepMesh_IncrementalMesh` has been protected against generation of exception with the shapes having a void bounding box.
- The last point is now preserved in the distributions created for B-spline curves by classes `GCPnts_UniformDeflection` and `GCPnts_QuasiUniformDeflection` with user-specified parameters `U1` and `U2`.
- Some corrections proposed by Dr. Paulos J. Nyirenda have been implemented:
 - Use of uninitialized local variables in `Approx_ComputeLine::ComputeCurve()` has been fixed;
 - Type of field "status" of the class `IntRes2d_Domain` has been converted from `Standard_Boolean` to `Standard_Integer`;
 - Methods `Geom2dAdaptor::MakeCurve()` and `GeomAdaptor::MakeCurve()` have been protected against null input curve.
- Earlier when a shape was copied by `BRepBulderAPI_Copy`, both topological (shapes) and geometrical (curves, surfaces) object copies were created.

It has become possible to copy only the topology, keeping the same geometry. This permits to:

- Save memory, avoiding replication of geometry if it is not required to change it;
- Keep association between other shapes and geometry of the copied shape (e.g. pcurves can be used to keep representation of the edge produced by intersection or projection on the shape)
- The algorithm of conversion of 2d curves into degenerated edges has been corrected.
- It has become possible to build a surface only on point constraints using class `GeomPlate_BuilderPlateSurface`.
- The class `Geom2dGcc_Circ2d3Tan` has been fixed to produce correct solutions
- The problem with absence of initialization of some fields in constructors of the class `BRepExtrema_SolutionElem` has been fixed.
- Static variables for local objects have been removed from the methods of the class `IntCurvesFace_Intersection` to avoid problems in multithreaded applications.
- Class `BRepProj_Projection` has been improved:
 - The class interface has become more intuitive;
 - The unintended side effect of adding pcurves created on temporary faces to original and resulting wires is now avoided.
- Class `BRepProj_Projector` has been improved to work more robustly in some specific cases.

Visualization

- Support of freetype and ftgl external libraries for advanced 2D and 3D text visualization has been introduced in OCCT. For this the architecture of visualization algorithms and other auxiliary functionalities has been revised, becoming more clear and flexible.

The main new features and improvements are as follows:

- Support of system fonts has been implemented. The library parses system configuration and searches all system fonts to use this information for the text 2D and 3D presentations.
- A virtual method `::DefaultTextHeight()` has been implemented in `Graphic3d_GraphicDriver` class to return the adopted text (or font) height parameter of `Prs3d` presentation for the current Graphic Driver. It is possible to use '-1' for default text height, instead of providing text height by the application.
- A new text output representation implemented in OCCT text drawing allows using system fonts alternatively to texture mapped fonts (WNT) or traditional bitmap fonts (UNIX, SUN). This means that now text output depends on the usage of FTGL and FreeType libraries.
- It has become possible to control basic Font properties (`FontName`, `FontAspect`, `Height`, `Align`, `Incline`, `Color`, `Zoomable`). If the user introduces a `FontName`, not present in the System font list, the algorithm returns the default "Courier" font. `Zoomable` permits to increase or decrease font (as object). `Incline` (in degrees) visualizes sloping characters. `Align` displaces the characters.

From the technical point of view:

- `LayerItems` have been implemented for `Visual3d_Layers`. This is a new approach to the visualization of 2D layer presentations. Each Item has two typical for presentation virtual methods: `ComputeLayerPrs` and `RedrawLayerPrs`. `ComputeLayerPrs` is called manually or before `RedrawLayerPrs` if `IsNeedToRecompute` flag is true. This approach to the visualization of Layer content helps to remove limitation of OpenGL lists (used in Layers by default). The old functionality is not changed.
- OpenGL package is now compiled by C++ compiler. The implementation of memory management and stack of graphic structures have been redesigned.
- All duplicates of `OpenGL_GraphicDriver` functionality have been removed as obsolete from `Graphic3d_GraphicDriver`.
- `InterfaceGraphic` package has been redesigned.
- The command: `vdrawtext`, which controls Base font functionality, has been added to DRAW Test Harness.
- New interactive object class `AIS_Triangulation` has been introduced in AIS package.

Its role is displaying triangulation encapsulated in `Poly_Triangulation` instance in a 3D view. The triangulation can be built by an application or by OCCT meshing tools. Optionally, individual colors for each triangulation vertex can be specified (for efficiency, each color is packed as a 32-bit integer, instead of using `Quantity_Color` objects).

The possibility to draw arrays of primitives (see `Graphic3d_ArrayOfPrimitives` class and the derived class hierarchy) with help of OpenGL Vertex Buffer Objects (VBO) has been added. This makes visualization of large primitive arrays several times faster compared to the "classic" implementation.

Higher performance is achieved by copying all primitive array data to the graphic card memory before drawing a primitive array for the first time. After this the data copy is deleted from RAM and should not therefore be accessed by the application.

The amount of available graphic card memory imposes a limit on the maximum primitive data array size (maximum number of primitives to be drawn) displayed with help of VBO.

It should be noted that VBO usage can be disabled if necessary with help of `Graphi c3d_Graphi cDriver: :EnableVBO(Standard_Boolean)` method. By default, vertex buffer objects are used if supported by the graphic subsystem, otherwise the old implementation of primitive arrays is used.

New DRAW Test Harness command (`vdrawsphere`) can be used to evaluate the implementation of primitives arrays with VBO usage turned on or off.

New methods to specify vertex colors as an integer value and to improve memory usage efficiency in case of a large number of primitives have been added to `Graphi c3d_ArrayOfPrimi tives` class.

- The new implementation of gradient background style allows filling the viewer background with different colors. The viewer has eight filling variants: two variants for each viewer side (horizontal, vertical), two variants for both diagonals and four variants for each corner. If the application has an image as the background, the gradient filling is not used.

The command `vsetgradientbg` for gradient filling has been added in the DRAW Test Harness command list. This command allows filling the viewer background in a gradient mode.

- `MeshVS_Sensi tive mesh` now returns a correct status of Matches after entities detection. This allows eliminating excess Mesh presentations that should not be involved in highlight and selection process.
 - `MeshVS_DataSource: :GetDetectedEnti ties` by point (X,Y) with Tolerance has an additional parameter `DMin`. This is an out argument that should be initialized by actual tolerance of entities detection in `DataSource`.
 - `MeshVS_Sensi tiveMesh` now treats success status of `MeshVS_DataSource: :GetDetectedEnti ties` to return them as result of `:Matches`. This allows eliminating excess mesh presentation owners from highlighting and selection process.
- `Fit All` algorithm for presentation of objects in viewer has been revised to provide consistent behavior for point objects (objects with zero bounding box) by setting some bounding limits (+1 for each side).

Infinite lines now have some specific properties: their bounding borders are calculated as the own point in the center of a line. `Fit All` command now affects infinite lines.

- `TKOpenGL` has become linked with `gl 2ps` as a third-party shared library instead of including `gl2ps` sources into OCCT sources.
- Selection of shapes in OCCT viewer has been improved: now depth and `DMin` parameters are computed more precisely in classes `Select3D_SelectableTri angulation` and `Select3D_SelectableSegment`. This allows avoiding confusing behavior of selection by mouse click when a wrong object could be selected if it had multiple points located under the mouse cursor at different depth.

Several new methods have been added in classes of `Select3D` package allowing access to the geometrical data of the currently selected (or highlighted) object under cursor. This can be used e.g. to provide user display of the point coordinates under the mouse cursor.

- The algorithm of choice of one among several possible overlapping objects selected by mouse click has been improved. Earlier the objects were selected by shape priority and, for example, a solid could not be selected because an overlapping shell had a greater priority.

Now the logic has been made more appropriate and the objects are first classified by depth, and then by other criteria (priority and distance from the mouse cursor), which are applied only to the objects closest to the user. To provide a behavior compatible with the previous implementation,

method `SetPickClosest()` of the instance of the class `SelectMgr_ViewerSelector` associated with the viewer should be called with argument `Standard_False`.

- The code used for polygonization of curves and for implementation of selection in the viewer has been synchronized with the code used for display. Otherwise the difference between selection and drawing might become critical, especially for long curves with high curvature.
- A new method `Graphic3d_Group::IsGroupPrimitivesAspectSet()` has been added to read aspects defined for a group to avoid errors while updating aspects in an existing `Graphic3d_Group`.
- Two improvements have been made in `OpenGL` package:
 - It has become possible to use `AVIWriter` in a multi-window application, as now it can distinguish window for recording;
 - Export functions providing access to GL extensions: `GLboolean OpenGL_QueryExtensionGLX(const char *extName)` and `GLboolean OpenGL_QueryExtension(const char *extName)` have been added.
- The bug in the method `Visual3d_View::SetViewMapping`, which incorrectly initialized projection matrix, has been corrected.
- GDI memory leak during export of `V3d_View` contents to enhanced metafile format (EMF) on Win32 platform has been corrected. The memory leak was really critical when a view contained `AIS_TextureShape` objects.
- A variable-length string class has been implemented to avoid problems with `CASROOT` path length.
- On Windows, display list sharing between several views has been corrected, so that display lists for user-defined markers could be used by several views without troubles. Besides a missing `glPixelStore()` call has been added to prevent user-defined marker shape corruption.
- To avoid exception caused by the time-consuming presentation manager 3D on a slow computer, a check of return value of the call to `TransientManagerBeginDraw()` has been introduced.
- `AIS_InteractiveContext::ClearGlobal()` internal method now removes the interactive object from the selection manager. This method is called by `Remove()` method, and the old code often resulted in memory leaks and forced developers to remove interactive objects from the selection manager at the application level.

Additionally, interactive objects are no longer erased to the collector viewer by default by `Erase()`, `EraseAll()` and `EraseSelected()` methods. `AIS_InteractiveContext::DisplayAll()` method also does not display objects from collector by default. This reflects the common trend to avoid using the collector viewer in applications.

- `MeshVS_ElementalColorPrsBuilder` class now works correctly if different front and back colors are specified for each mesh element while `MeshVS_DA_ColorReflection` attribute is set to false. A new `DRAW TestHarness` command (`meshcolors`) has been introduced to test various ways of displaying scalar data on a mesh with colors.
- The fact that on some platforms (SUN, IRIX) the font name starts in a lower case has been taken into consideration.
- The behavior of `PrsMgr_PresentationManager::Clear(anObject, aMode)` method has changed. Now this method destroys the corresponding `PrsMgr_Presentation` instance instead of just clearing its contents. Keeping such "cleared" presentations previously resulted in some inconsistencies in the behavior of `AIS` and `OpenGL` packages.

After the correction, it has become possible to call `AIS_InteractiveContext::Display()` method for an object removed from the context with the help of `AIS_InteractiveContext::Remove()` without re-creating the interactive object instance.

Application Framework

- New TObj resource file has been added to StdResource package in order to provide TObj persistence. Besides two new variables have been added to OCCT environment: CSF_TObj Resources and CSF_TObj Defaults.
- The non regressive testing of naming mechanism has been implemented into OCCT. The content of DNaming package has been extended with a special group of "Modeling Commands" which allow creating an OCAF-based data framework having all necessary information for the performing the services of the Naming component algorithm.

The final commands which perform Naming testing of the resulting shape are:

- TestSingleSelection - checks correct selection of each sub-shape of the resulting shape
- TestMultipleSelection - all sub-shapes of the same type (Faces, Edges ...) are put in Compound and the Compound is Named.
- The methods `::Depth()`, `::IsDescendent()` and `::Root()` now work faster because the pointers are called directly without Handle-like smart-pointers.
- A problem with documents containing a single null shape saved in wrong way so that they could not be loaded back has been fixed in OCAF binary persistence. The writing has been corrected to produce a valid file, and the reading has been improved to recognize and properly process wrong files.
- The method `TDataStd_BooleanArray::SetValue()` now properly works with FALSE argument.

Data Exchange

- The possibility to convert elementary surfaces (cylindrical, conical or spherical) to BSpline surfaces with required continuity, degree and number of spans has been introduced. Conversion of elementary surfaces is managed by the resource file.
- VRML reader now works faster due to the improved method `VrmlData_IndexedFaceSet::TShape()`.
- SplitContinuity operator has been improved to split pcurves by C2 criterion.
- The classes of StepToGeom package have been modified to improve the performance. For each class of this package:
 - inheritance from StepToGeom_Root is eliminated;
 - constructor, `IsDone` and `Value` methods are replaced by a single `Convert` method of the class.
- The algorithms have been improved to remove very small edges with length less than parameter `MinTolerance()`.
- The level of optimization has been changed from O2 to O1 for StepAP214, StepToGeom and RWStepAP214 packages of TKSTEP toolkit.
- ShapeCustom_BsplineRestriction has been adapted to convert various types of elementary surfaces
- Useful modifications to the source code contributed by Datakit have been implemented.

Draw

- Two improvements are made in TKDraw packages:
 - Message: :DefaultMessenger() is initialized by default so as to have sent messages shown in Draw console
 - Code corrected to avoid linker warnings on Windows (MSVC++ compiler).
- Dump method has been added in the class Draw_Segment2D so that the command 'dump' could print the end point coordinates of the corresponding object correctly. It can be convenient to create such objects inside the algorithm to inspect them later in the Draw command line.
- It has become possible to put both NIS and AIS interactive objects in the internal data map of names and objects. Furthermore, the option to show the error status of recording AVI file in the command vrecord has been introduced.
- New command VDrawSphere X Y Z Fineness Radius EnableVBO [NumberOfViewerUpdate(Default=1)] has been implemented to replace Visualization Performance Meter, which has been removed from OCCT. It allows building a sphere with given coordinates, radius and fineness and then displays the information about the properties of the sphere, the time and the amount of memory required to build it. This command is used for visualization performance evaluation.

Documentation

- The documentation generation procedure has been completely revised and the latest Doxygen version 1.6.1 has been implemented. Earlier Doxygen versions, for example, 1.5.3 cannot process the documentation correctly.

The main improvements are as follows:

- All documentation is generated at a single execution of Doxygen.
- Complex post-processing of generated files has been replaced by equivalent code provided to Doxygen on input, as a result, the size of scripts has decreased by many times;
- Cross-references are generated for all OCCT classes
- OCCT version number is included into documentation
- The time of execution has been reduced (~30 min on PC workstation for whole OCCT)
- Complete in-browser search by all classes (as provided by Doxygen 1.6.x)
- No Standard_EXPORT entries in documentation
- Possibility to generate documentation for the selected list of modules (note that the possibility to generate the documentation for a toolkit or a package has been dismissed)
- Possibility to generate documentation for Products (optionally with cross-references to OCCT docs, using tag files)

The generation procedure has also changed; the simplest way is:

```
WOK: wb> source [woklocate -p WOKToolLib:source:OCCTDocumentation.tcl]
```

```
WOK: wb> doxygen [OCCTDoc_MakeDoxyfile <output_dir>]
```





The command OCCDoc_MakeDoxyfile has options to define a sub-set of modules to process, specify the path to Graphviz/bin directory, set the search option (YES/NO), and specify the location of tag file for external references from product to OCCT.

WOK

- MS Visual Studio projects generator in WOK has been updated to generate projects for MS Visual Studio 2010.
- The problem with generation of vcpr objects for several executables has been fixed.

Dependencies and Packaging

3rd Party Products

- Starting from OCCT 6.4, FLTK products have been eliminated from dependencies and binary distribution package.

The new versions of other products are used since now; see the following table for details.

<u>Products</u>	<u>Description</u>
freetype-2.3.7 ftgl-2.1.2 gl2ps-1.3.5	New products. Mandatory products for OCCT building and starting DRAW, samples and demos.
tbb30_018oss	New products. Optional products for OCCT building and starting DRAW, samples and demos.
tcltk-8.5.8	tcltk 8.4.14 has been replaced by 8.5.8
qt-4.6.2	qt 4.3.3 has been replaced by 4.6.2
java-1.6.0 (build 20)	jdk 1.3.1 has been replaced by 1.6.20

Packaging

- WOK tool has been removed from OCCT package.
Starting from OCCT 6.4 WOK is distributed in source and binary form as independent packages.
These packages are available without special restrictions.
- DFBrowser tool has been removed from OCCT delivery.
Starting from OCCT 6.4 QT DFBrowser is delivered as an independent OCCT tool available only to the customers.



Building Tools

- Starting from OCCT 6.4 project files for MS Visual Studio 2003 are not provided.
- The structure of MS Visual Studio solution OCCT.sln file for (re)building a complete set of OCCT libraries in one instance has been modified. It has become possible to build OCCT modules separately.
- MS Visual Studio projects are now provided for Visual Studio 2005, 2008 and 2010 in 32 and 64 mode
- MS Visual Studio projects generate binaries in subfolders vc8, vc9 and vc10 of folders win32 and win64. New generation structure:
 - ros
 - win32
 - vc8
 - vc9
 - vc10
 - win64
 - vc8
 - vc9
 - vc10
- New options of Makefiles provide for configuration of OCCT 6.4 with TBB, freetype, ftgl, gl2ps and qt products correspondingly:
 - -with-tbb-include and -with-tbb-library;
 - --with-freetype;
 - --with-ftgl;
 - --with-gl2ps;
 - --with-qt;

Binary package

- Starting from OCCT 6.4 binary packages are built by MS Visual Studio 2005 SP1 32-bit.

Porting from OCCT 6.3.1

Porting of user applications from the previous OCCT version to version 6.4 requires the following major issues to be taken into account:

- Additional dependencies appeared on third-party libraries (freetype2, ftgl, gl2ps, tbb, tcl, etc.). Download these libraries from their respective websites or from dedicated page provided by OPEN CASCADE for supported platforms. Make sure to set up the necessary environment for loading these libraries.
- If you meet problems installing or using Intel(R) TBB library and running parallelized version of BRepMesh, disable TBB using the options of build tools
- Binaries on Windows platform have been put into sub-folders corresponding to Visual Studio version; this needs to be taken into account when building and running dependent applications
- Use of VBO (Vertex Buffer Objects) in 3d viewer can require updating the graphics card driver to work correctly. If it is not possible to update the driver, consider disabling use of VBO.
- Global macros `GetSStrng()` has disappeared. If it was used in the code of your application, revise the code for direct use of the argument stream object.
- You might need to revise the code related to text display in 3d viewer to take into account new approach of using system fonts via `ftgl` library.
- You might experience difference in tolerances of shapes produced by application algorithms due to changed behavior of `BRepTools_Modifier` class. In such case, identify and fix the problem in the algorithm.
- If methods `GCPnts_QuasiUniformDeflection` and `GCPnts_UniformDeflection` have been used with workarounds against previous inconsistent behavior, such workarounds need to be removed.
- If improved objects selection logics in 3d viewer is not suitable for your application, consider activating compatibility mode by call to method `SelectMgr_ViewerSelector::SetPickClosest()` with `Standard_False`.
- The code manipulating AIS objects in 3d viewer might need to be revised due to changed behavior of `Erase()` and `Remove()` methods of `AIS_InteractiveContext` class.

See the text above for more details on relevant changes.

Products

ACIS SAT Interface

- New methods ReadStream() and WriteStream() allowing to read and write SAT data to and from streams have been added to the relevant API classes (SatControl_Reader, SatControl_Writer, SATCAFControl_Reader, SATCAFControl_Writer).

New commands to test the functionality have been added to DRAW Test Harness:

- satreadstream filename shape - creates an input stream from the given SAT file and converts data to shapes.
- ReadSatStream Doc filename - creates an input stream from the given SAT file and converts data to XCAF document.
- WriteSatStream Doc filename - creates an output stream from the given file and writes XCAF document to it.

OMF

- Boolean Operations have been extended with new types of operations appended to the definition of the enumeration SMDSBool_Operation:
 - SMDSBool_SCOMMON : finds common between two arguments where the first is a surface and the second is a solid,
 - SMDSBool_SCOMMON21 : finds common between two arguments where the first is a solid and the second is a surface,
 - SMDSBool_SCUT : finds cut between two arguments where the first is a surface and the second is a solid,
 - SMDSBool_SCUT21 : finds cut between two arguments where the first is a solid and the second is a surface.
- The tools providing possibility to read mesh with colors and texture from OBJ file and display it in 3d viewer have been introduced in SMDSTools and SMDSI nter packages.

DXF

- The ACIS entities of version 21500 are now supported and can be read.
- The handling of color index information by XCAF translator has been corrected.

BestFit

- A new product called "BestFit" has been added in the gallery of OCC Products.

BestFit transforms a cloud of 3d points in relation to a given shape to minimize the normal distance between the points and the shape.

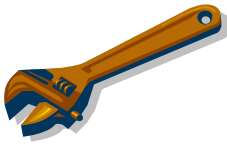
The product is composed of three modules:

- - MBFIT - BestFit library, the main algorithmic library of the product.
- - BFITDraw - Draw plugin to test BestFit product.
- - BFITSample - MFC-based BestFit sample application.





Bug Fixes



- Since the last minor release (version 6.3.1) Open CASCADE Technology and Products ver. 6.4 incorporates **149** modifications (bug fixes, enhancements and other corrections). For details, refer to the [Appendix](#).



Appendix: Modifications in Open CASCADE Technology and Products version 6.4

- o [Foundation Classes](#)
- o [Modeling Data](#)
- o [Modeling Algorithms](#)
- o [Visualization](#)
- o [Application Framework](#)
- o [Data Exchange](#)
- o [WOK](#)
- o [Draw](#)
- o [Documentation](#)

Products

- o [OMF](#)
- o [ACIS SAT Interface](#)
- o [DXF](#)
- o [BestFit](#)

Foundation Classes, 25 modifications	
ID	Short Description
18768	Make Ncol l e c t i o n containers use an allocator (where it has not been done yet)
20300	Uninitialized member variables in gp package
20728	An additional OSD_Thread: : Wai t () method
21446	New algorithm to find minimal loops on a graph
21457	Make Standard: : Purge () work better to return more memory to the system
21537	OSD_Path class loses up-directory entry ("..") in paths ending with "../.name"
21552	Minor improvements in NCol l e c t i o n package
21556	Add the method Poly: : Catenate to merge several triangulations into one object
21592	OSD_Path loses path separator (slash) in paths containing one-letter directory and file name
21669	Invalid trunk operation in OSD_FontMgr. cxx
21711	Assignment operator for maps in Ncol l e c t i o n unnecessarily increases buckets array
21713	Incorrect decimal separator in GL2PS export on Linux
21723	Few minor improvements in Standard package
21724	OSD_Chronometer works inconsistently on Windows and Linux in case of multithreaded application
21731	Improvement of Message_ProgressSentry interface
21735	Method gp_Trsf: : GetRotati o n () does not work for zero rotation
21758	Dump of "maxtolerance" command contains unexpected symbols.
21783	Add the global function ACosApprox for quick calculation of the approximate value of the arc cosine of a real argument
21784	Make NCol l e c t i o n_I ncAl l o c a t o r raise exception if no memory available
21785	Derive from NCol l e c t i o n_Ba s eAl l o c a t o r the class that uses the global dynamic heap (malloc / free)
21789	Add methods to convert Quanti ty_Col o r to integer ARGB value and vice versa
21790	Forbid copying of Sentry object that provides interface to Standard_Mutex



21809	Implementation of quaternions for smooth 3d rotations
21830	Bug in Message_ProgressIndicator when working with infinite scale
21913	Minor corrections in various classes (virtual methods, protections)
Modeling Data, 1 modification	
ID	Short Description
21546	Classes Extrema_GenLocateExt... can raise exceptions if initial point is on parametric limit, due to rounding errors
Modeling Algorithms, 57 modifications	
ID	Short Description
20285	SIGFPE Arithmetic exception on MakeCut
20404	Command distance gives wrong result for given shapes
20822	Bug in BRepOffsetAPI_MakeOffset: 2d offset algorithm crashes on some wires containing only segments and arcs.
20829	Problem in MOA on Mandriva 2008 in opt mode
21179	BRepTools_NurbsConvertModification does not convert 2d curve of a degenerated edge
21184	Class ShapeAnalysis_FreeBounds can not create wire from connected edges
21208	Correction for FIP "BO via partition"
21235	Fixes and improvements powering ShapeUpgrade package.
21255	Some problems with 3D chamfer construction
21261	Implementation of thickening of shell functionality
21307	Elimination of redundant templates
21403	Operator SplitContinuity does not perform splitting pcurves by C2 criterion
21448	Invalid result of BRepOffsetAPI_MakePipeShell
21499	ShapeAnalysis_FreeBounds::ConnectEdgesToWires() builds wrong wires
21616	ShapeAnalysis_FreeBounds problem
21623	Operation of removing small edges in the class ShapeFix_Shape does not work.
21510	Problem of DRAWEXE launching on SunOS 5.1 compiled in optimize mode
21518	Improvement of the function Angle2D in BOP_WireSplitter.cxx
21521	Invalid result DRAW command "segsur" on surface obtained from attached face.
21534	Need correction of IntTools_Context (attached) for Salome partition algorithm
21540	Boolean operation section produces wrong result (OCC 6.3.1)
21547	Enable possibility to copy shape keeping same geometry in BRepBuilderAPI_Copy
21548	Problem with cut operation
21553	Possibility to convert plane surfaces to NURBS in ShapeCustom
21562	Bug in class FEmTool_LinearFixation
21568	Invalid work of ExtremaCurveCurve
21572	The bug is appendix to the Salome Bug 0020642
21588	Appendix to the SALOME bug: 0020683
21594	BRepTools_Modifier increases tolerances of subshapes unreasonably
21599	Class GeomPlate_BuildPlateSurface does not build surface by points constraints if initial plane is specified.
21603	The bug is appendix to the Salome Bug 0020707
21614	Geom2dGcc_Circ2d3Tan does not compute all possible solutions
21618	Appendix to the SALOME bug: 0020730
21625	Projection of edge on face yields empty result



21638	The bug is appendix to the Salome Bug 0020773, 0020774
21642	BRepExtrema_DistanceSS returns swapped solutions when second shape contains edges on C0 curves
21652	Problem with offset on the wire
21691	Invalid result of bcut operation
21645	Slow and bad triangulation on attached shape.
21737	Adaptation of ShapeCustom_BSplineRestriction to convert various types elementary surfaces
21766	Sewing produce invalid result on attached shape.
21771	Method ShapeAnalysis_FreeBounds::ConnectWiresToWires can not build one wire from a few connected wires
21775	Exception is raised during sewing shape in non-manifold mode.
21969	Sewing modified initial shape for case when sewing was not produced
21770	Exception in intersection algorithms (projection of edge on face)
21772	BrepTools_Modifier breaks shape containing seam-like edge
21780	GCPnts_QuasiUniformDeflection on unspecified behavior
21781	Exception in Incremental Mesh with shapes having void bounding box
21799	BOP Invalid Cases.
21810	Uninitialized fields in BrepExtrema_SolutionElem class
21811	Eliminate unnecessary static variables in IntCurvesFace_Intersection
21861	GCPnts_UniformDeflection on lost last point
21882	BOP: Bad Tolerance.
21892	Corrections proposed by Dr. Paulos J. Nyirenda
21914	Projection of edge on face creates edge inside inner hole
21921	Corrections in BRepProj_Projection class
21961	Paralleling BRepMesh using Intel TBB tools
Visualization, 22 modifications	
ID	Short Description
21091	Text rendering based on FTGL library
21171	Problem with vfit of vertex and edge
21209	GDI memory leak upon exporting a 3D view to EMF
21262	Appendix for Mantis bug 0020426.
21332	Corrupted user-defined marker in multi-view applications
21450	Missing text in PDF export
21486	Use vertex buffer objects to draw arrays of primitives
21492	The presentation manager 3D may crash the application on slow computer
21522	Graphic3d_Group group aspects state
21567	Shading problem
21639	AI S_Interactive object cannot be displayed after Remove from IC
21656	Incorrect shape can be selected in the viewer in case of overlapped shapes
21658	Improve logic of selection by mouse click in 3d viewer
21660	Advanced selection in MeshVS with several presentations does not work properly
21671	Removed presentation still selectable in View3D
21720	MeshVS: Double-colored mesh elements are drawn incorrectly
21732	vi nit command displays 3D Viewer but work is stopped by DRAW interpreter
21747	Implementation of gradient background style
21763	SelectMgr_ViewSelector returns wrong selectable entity for the selected owner
21774	Selection mismatch representation

21786	OpenGL improvements (AVI Writer, GL extensions)
21791	Bug in the method Visual3d_View::SetViewMapping
Application Framework, 14 modifications	
ID	Short Description
20795	Several methods of TDataStd_TreeNode might be accelerated.
21231	Missing resource necessary for TObj_persistence
21424	Naming fails for the specified case (3 revolutions & 2 Fuse)
21519	A mistake in TDataStd_BooleanArray::SetValue(false)
21542	Implementation of naming mechanism for non regressive testing
21551	OCAF fails to read file containing null shape
21622	Naming of a single edge creates Compound of edges
21630	Naming of any of four vertexes presented on the picture gives Compound of vertexes instead of a single vertex
21635	Naming of main part of faces of the presented shape fails
21636	Naming of the face #14 of the presented mode raises NullObject exception
21651	Recomputation of Naming data structure fails for several faces (and wires of these faces)
21653	Naming fails during Wire of Sphere naming
21662	Selection of a face generated by Fillet function hangs PC
21673	Selection of Edge gives compound of edges
Data Exchange, 13 modifications	
ID	Short Description
20812	Crash (stack overflow) on writing STEP file
21131	IGES reader: surfaces remain untrimmed while boundary edges exist.
21149	Message_Alogarithm.Ixx has missing includes
21232	Shape was not sewed completely with any tolerance.
21347	Optimization of Vrml reader
21385	Problem with recursive binders
21404	Crash on reading STEP file
21487	Entities having invisible status in IGES file did not have invisible status after reading with command ReadIgesi n XDE document in DRAW Test Harness
21609	Bad result after re-writing to IGES.
21617	Sub-shapes from result shape are not contained in the history map after shape processing
21683	Blocked problem. TKSTEP toolkit is not built on SGI FUEL platform
21706	Errors during jni extraction in units: STEPConstruct & STEPCAFControl
21924	New development and improvement data translators
WOK, 2 modifications	
ID	Short Description
21181	Update of generator of Visual Studio projects.
21929	Improvement of VC projects generation by WOK



DRAW, 4 modifications	
ID	Short Description
21634	Minor improvements in TKDraw
21684	DRAW compiled with tcl tk 8.5 is crashed during launching
21782	Dump command does not work for Draw_Segment2D
21794	Improvements in ViewerTest (visualization in Draw)
Documentation, 2 modifications	
ID	Short Description
21488	Wrong documentation in BRepBuilderAPI_MakeFace algorithm
21674	Improve generator of reference documentation

Product Bug Fixes

The following bug fixes have been performed for OPEN CASCADE Products customers.

OMF, 2 modifications	
ID	Short Description
21193	OBJ file reader & displayer
21459	New types of Boolean operations
ACIS SAT Interface, 1 modification	
ID	Short Description
21661	Add API to work with streams
DXF, 5 modifications	
ID	Short Description
21042	Patch for DXF reader
21491	Crash reading DXF file
21602	Black color is not correctly encoded during export to DXF
21621	Export attached shape in DXF without using ACIS entities is failed.
21768	ACIS solids are not imported
BestFit, 1 modification	
ID	Short Description
21490	Creation of BestFit product

