

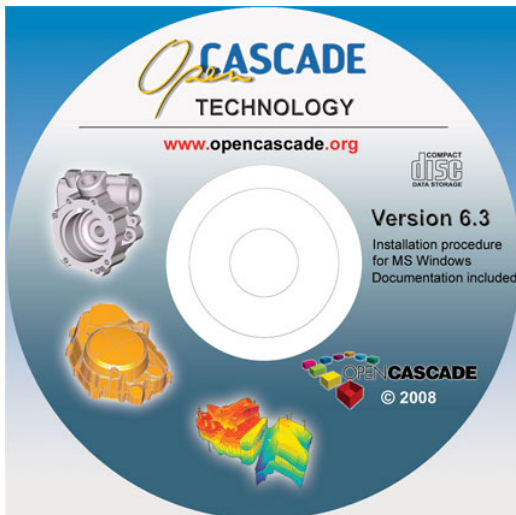


## Open CASCADE 6.3.1 Maintenance Release

### Release Notes

#### Overview

**Open CASCADE Technology 6.3.1** is a maintenance release, which includes new features, improvements and bug fixes, over minor public release 6.3. This release is available exclusively to OPEN CASCADE customers.



Version **6.3.1** is 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 6.3.1.



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## Highlights

### Open CASCADE Technology

- A number of improvements has been made in Foundation Classes and Modeling libraries to facilitate the usage of OCCT in multithreaded applications
- A mechanism to collect statistics and debug memory allocation with OCCT optimized memory manager has been implemented
- A basic OCAF toolkit (TKLCAF) has become independent from OCCT modeling libraries
- Topological naming in OCAF has been considerably improved
- Geometry construction algorithms have been extended by advanced approximation methods
- Numerous improvements and bug fixes suggested by OCCT users have been implemented in Visualization libraries
- New supported compiler: Visual C++ 9.0 (Visual Studio 2008)

### Products

- Products have been extended by DRAW Test Harness plug-ins providing commands useful for testing and initial acquaintance with these libraries.



## New Features

### Foundation Classes

- Memory management in mode MMGT\_OPT=1 has been extended with a mechanism that allows to see memory usage statistics when debugging memory-related problems.

This mechanism is based on calling a user callback function from `Allocate` and `Free` methods of the class `Standard_MMGrOpt`. These methods pass the current operation type, the block address and its size to that function. The user can define the callback with the method `Standard_MMGrOpt::SetCallbackFunction`. By default no callback is defined.

The class `NCollection_BaseAllocator` provides the static method `StandardCallback` that can be used as a callback function.

The static method `NCollection_BaseAllocator::PrintMemUsageStatistics` can be used to print memory usage statistics accumulated by `StandardCallback` into standard output.

- New template class `NCollection_Handle` is added. It can be used to define a Handle adaptor for dynamically allocated objects of arbitrary type. The advantage is that this handle will automatically destroy the object when the last referred Handle is destroyed (i.e. it is a typical smart pointer), and that it can be handled as `Handle(Standard_Transient)` in OCCT components.
- New header file `Standard_DefineException.hxx` provides pre-processor macros to facilitate definition of new exception classes conforming to OCCT requirements (in the same way as macros provided by `Standard_DefineHandle.hxx` for definition of transient classes).

### Modeling Algorithms

- Advanced approximation method (implemented in class `AppDef_TheVariational`) has been included in algorithms for creation of curves and surfaces:
  - `GeomAPI_PointsToBSpline`,
  - `GeomAPI_PointsToBSplineSurface`,
  - `Geom2dAPI_PointsToBSpline`,
  - `GeomFill_AppSurf`,
  - `BRepOffsetAPI_ThruSection`

The approximation method can be chosen according to the new parameter added to relevant methods (`Approx_ParametriizationType`). See the documentation on the modified classes for details.

### Application Framework

- A possibility to take shape orientation into account during its Selection / Regeneration has been added. To implement it the `Select` method of `TNaming_Selector` class has received a new Boolean parameter `Orientation` and `TNaming_Name` class - `new name = TNaming_ORIENTATION`. By default this parameter has value=False and shape orientation is ignored. If `Orientation = True`, the selected shape is regenerated using the Naming solver taking orientation into account.



- Quality and safety of the Topological naming implemented by TNaming package have been improved. Now the Naming mechanism allows to:
  - keep (if necessary) the selected shape Orientation;
  - guarantee that the selected shape belongs to the Context;
  - process all aggregation shapes (Compounds, Shells and etc.);
  - extend data definition if the selected shape is a Wire;
  - correct computation of results after a multiple selection.
- Persistence has been implemented for OCAF attributes used for storing Dimension tolerances and materials in XDE documents.

### Building Tools

- Additional MS Visual Studio solution file OCCT.sln has been provided along with the relevant project files for VC++ versions 7.1, 8.0, 9.0. This solution comprises all OCCT projects and can be used for (re)building a complete set of OCCT libraries in one instance of Visual Studio.

### Products

- Structure and contents of Product packages have been revised and became more uniform.
- Product packages have been extended by DRAW Test Harness plug-ins providing commands useful for testing and initial acquaintance with these libraries. The plug-ins and provided commands are described in updated User's Guides on the Products:
  - Parasolid
  - Canonical Recognition
  - DXF
  - Express Mesh
  - Open CASCADE Mesh Framework (OMF)
  - ACIS / SAT
  - Surfaces from Scattered Points (SSP)





## Improvements

### Foundation Classes

- In the `Units` package, conversion of thermal parameter values from SI to user system units and vice versa has been fixed.
- A bug that led to incorrect work with negative keys has been fixed in the class `TColStd_PackedMapOfInteger`.
- Instantiation of `OSD_Host` class has been removed from `IGESData`, thus eliminating a confusing situation when IGES translator could try to access network facilities and trigger security warnings.
- Implementation of Open CASCADE RTTI has become more thread-safe through initialisation of relevant static objects (`STANDARD_TYPE`) at start-up.
- The functions for evaluation of B-splines (`BSplCLib` and `BSplSLib` packages) and approximation (`AdvApprox` and others) have been revised to avoid usage of static data, which aims to improve thread safety of these libraries.

### Modeling Algorithms

- The performance of the algorithm searching extreme points for curves and surfaces has been improved for some cases (mainly for non-analytical curves and surfaces).
- A fix has been implemented to avoid large tolerance after sewing, caused by invalid cutting of curves.
- Method `IntTools_BeanFaceIntersector::Perform()` has been modified to be coherent with the source data of B-Spline surfaces.
- A fix has been implemented to avoid invalid result of method `BSplCLib::RemoveKnot()` in the case when B-spline knot multiplicity is equal to its degree
- `TopAbs` and `TopoDS` packages have been improved to make most of simple methods inline and to use more effective algorithms based on tables / arrays (instead of switches)
- Classes `LocOpe_SplitShape.cxx` and `LocOpe_WiresOnShape.cxx` have been modified to avoid exception during shape splitting by a tangential wire.
- Method `BRep_Tool::IsClipped()` has been corrected to prevent returning `True` for the cases when the basic surface is a plane.
- The algorithm processing closed intersection curves in Boolean operations has been improved.
- The algorithm of computation of shrunk ranges for the edges that are based on lines has been improved.
- `BRepMesh` algorithm has been improved to check more consistently in case of wires intersection (using mesh tolerance) and classify points avoiding some triangulation errors.

- Regression caused by incorrect handling of internal edges in triangulation algorithm (occurred in OCCT 6.3.0) has been fixed.
- A number of corrections and improvements have been made in MeshDS, MeshAI go, and BRepMesh packages for optimization of triangulation algorithms. Domain flag and relevant data have been removed from mesh data structures to decrease memory allocation; unused package MeshShape and several obsolete classes have been removed.
- Memory leak in BRepMesh (with BRepMesh\_Cl assi fi er object, occurred in OCCT 6.3.0) has been fixed.
- Classes ShapeAnal ysi s\_FreeBounds and ShapeAnal ysi s\_BoxBndTree have been modified to fix bug of creation of wires from edges with non-shared vertices.

### Visualization

- Several corrections suggested by Open CASCADE Community contributors have been implemented. They resolve some low-level visualization issues mostly in OpenGL package:
  - Marker positioning in a 3D view has become more precise, which is achieved by floating-point type usage in calculations.
  - It has become possible to invoke a 3D view callback function both before and after redrawing the overlayer. The callback function can understand whether it is called before the overlayer by checking OCC\_PRE\_OVERLAY.
  - The code that clears the frame buffer at the beginning of redrawing of a 3D scene has been corrected to avoid flicker.
  - The problem with clipping planes cutting off pieces of the trihedron has been resolved.
  - The state of lighting is no longer stored in a variable; instead it is always queried through an OpenGL call to avoid desynchronizing.
  - The z-buffered trihedron has been corrected in order to avoid spontaneous color change.

Apart from that, the problem with continuous texture-mapped fonts reloading in multi-view applications has been corrected, so that the textures are no longer reloaded for the same font on each scene update.

- Documentation of Sel ect3D\_Sensi ti veFace: : ComputeDepth() method has been improved to suggest using custom sensitive entities at the application level in case of two parallel sensitive faces "almost parallel" to the view plane and having a very small gap between them. However, Sel ect3D\_Sensi ti veFace: : ComputeDepth() is simple, fast and suitable for most other cases.
- On Linux, the size of axis labels created by V3d\_Vi ew: : Tri edronDi spl ay() method in V3d\_ZBUFFER visualization type has been decreased to match it with the typical axis size. The "vzbufftri hedron" DRAW command has been added for displaying the trihedron in the test 3D view.
- A new FastConverter class has been introduced in the Voxel package. It converts a boundary represented shape (TopoDS\_Shape) into voxels (Voxel\_Boo l DS or Voxel\_Co l orDS). This class uses a very fast algorithm (it works much faster than Voxel\_Conver ter).



- The implementation of resizing of a 3D view window or the main application window in WNT\_Window class constructor has been modified to avoid the problem when the view is first filled with the widget background and only then a 3D scene is drawn.
- The limitation of a quantity of allocated V3d\_VIEWS (by 100) has been removed. A mechanism releasing the unallocated IDs of the views has been implemented.
- The method ChoosePixelFormat has been modified to correctly provide 24-bits visualization in the OCCT viewer on Intel 915 or 945 chipset.

### Application Framework

- The Index in NamingDriver of Binary format is now correctly nullified if Stop\_shape is null.
- Null handle arguments are now correctly processed to avoid the exception ACCESS\_VIOLATION and to have the possibility to unset string value of TObj\_Object.

### Data Exchange

- Missing function XCAFDoc\_DimTolTool::GetDatum() has been implemented.
- Color information has become supported by assembly components represented as compounds in STEP translator, after the bug in FindEntities static function in STEPCAFControl\_Writer.cxx has been fixed.
- Classes ShapeProcess\_ShapeContext and XSAIgo\_AIgoContainer have been modified to avoid loss of mapping between IGES entities and resulting shapes.

### Test Harness

- Check on the NULL object has been added in XShow command.

### WOK

- Generation of auxiliary C++ code for CDL classes has been revised to use macros provided by Standard\_DefineHandle.hxx and Standard\_DefineException.hxx instead of independently defined templates. Definitions of these macros have been revised for more clarity and consistency.
- Generation of BinPugin.vcproj has been corrected for Visual C++ 8.0 64 bit.
- A generator of MS Visual Studio project files has been updated to generate also projects for Visual C++ 9.0.



## Products

### ACIS Interface

- Construction of new types of curves (defined by ACIS laws) has been implemented in ACIS import.

### DXF

- The elements marked for deletion after fast discrete operation are now correctly removed to correctly export faces based on RectangularTrimmedSurface.

### Express Mesh

- It has become possible to mesh a face with several holes touching the external boundary after the respective bug has been fixed.

### OMF

- The bug in the algorithm of Boolean Operations on meshes that caused incorrect processing of intersection contours located entirely inside the boundaries of a face of the object mesh has been fixed.
- The robustness of the algorithm of Boolean Operations on meshes has been improved. Additionally, exhaustive grid tests to perform non-regression testing of the Boolean operations algorithm on meshes have been created.



## Changes

### Foundation Classes

- A lot of duplicate template instantiations have been removed. The following replacements should be made in applications that might have used removed classes:

Removed class	Class to be used
Xw_ListOfIndices	TColStd_Array1OfInteger
Xw_HListOfIndices	TColStd_HArray1OfInteger
Xw_ListOfMFTSizes	TShort_Array1OfShortReal
Xw_HListOfMFTSizes	TShort_HArray1OfShortReal
PlotMgt_ListOfMFTSizes	TShort_Array1OfShortReal
PlotMgt_HListOfMFTSizes	TShort_HArray1OfShortReal
WNT_ListOfMFTSizes	TShort_Array1OfShortReal
WNT_HListOfMFTSizes	TShort_HArray1OfShortReal
AI_S_DataMapOfTransientTransient	TColStd_DataMapOfTransientTransient
Transfer_DataMapOfTransientTransient	TColStd_DataMapOfTransientTransient
TDataStd_DataMapOfStringInteger	TColStd_DataMapOfStringInteger
TDataStd_HArray1OfByte	TColStd_HArray1OfByte
Approx_SequenceOfArray1OfPnt2d	TColgp_SequenceOfArray1OfPnt2d
AppBlend_SequenceOfArray1OfPnt2d	TColgp_SequenceOfArray1OfPnt2d
BRepMesh_DataMapOfShapeReal	TopTools_DataMapOfShapeReal
BRepOffset_DataMapOfShapeReal	TopTools_DataMapOfShapeReal
ShapeAnalysis_DataMapOfShapeReal	TopTools_DataMapOfShapeReal
TopOpeBRepTool_DataMapOfShapeReal	TopTools_DataMapOfShapeReal
TopOpeBRepBuild_DataMapOfShapeReal	TopTools_DataMapOfShapeReal
TopOpeBRepBuild_DataMapOfShapeInteger	TopTools_DataMapOfShapeInteger
TopOpeBRep_DataMapOfShapeInteger	TopTools_DataMapOfShapeInteger
TopOpeBRepTool_IndexedDataMapOfShapeAddress	TopTools_IndexedDataMapOfShapeAddress
IntTools_IndexedDataMapOfShapeAddress	TopTools_IndexedDataMapOfShapeAddress
MAT2d_SequenceOfBoolean	TColStd_SequenceOfBoolean
Extrema_SequenceOfBoolean	TColStd_SequenceOfBoolean
GraphiC3d_Array1OfBytes	TColStd_Array1OfByte
GraphiC3d_HArray1OfBytes	TColStd_HArray1OfByte
Units_StrgSequence	TColStd_SequenceOfHAsciiString
Units_StringsSequence	TColStd_HSequenceOfHAsciiString
GeomFill_SequenceOfCurve	TColGeom_SequenceOfCurve
MDocStd_DocEntryList	TColStd_ListOfHAsciiString
TestTopOpeDraw_ListOfHAsciiString	TColStd_ListOfHAsciiString
BinObjMgt_SequenceOfAddress	TColStd_SequenceOfAddress
SelectBasis_SequenceOfAddress	TColStd_SequenceOfAddress
GraphiC3d_SequenceOfAddress	TColStd_SequenceOfAddress
GGraphiC2d_SequenceOfCurve	TColGeom2d_SequenceOfCurve
BRepFill_DataMapOfShapeSequenceOfShape	TopTools_DataMapOfShapeSequenceOfShape
BRepOffsetAPI_DataMapOfShapeSequenceOfShape	TopTools_DataMapOfShapeSequenceOfShape
BRepMesh_MapOfHAsciiString	TColStd_MapOfHAsciiString
Storage_MapOfHAsciiString	TColStd_MapOfHAsciiString
Interface_DataMapOfIntegerTransient	TColStd_DataMapOfIntegerTransient
STEPCAFControl_DataMapOfShapeLabel	XCAFDoc_DataMapOfShapeLabel
STEPCAFControl_DataMapOfShapeTransient	MoniTool_DataMapOfShapeLabel
Resource_Array1OfHAsciiString	TColStd_Array1OfHAsciiString
CPPJini_DataMapOfHAsciiStringInteger	TColStd_DataMapOfHAsciiStringInteger



- Definition of type `AdvApprox_EvaluatorFunction` has changed: earlier it has been a pointer to C function, now it is an abstract class with a single method (having the same meaning and parameters as the function before). The code that uses tools from `AdvApprox` package might need to be revised: static functions and associated global variables used for feeding approximation algorithms should be encapsulated to a class inheriting `AdvApprox_EvaluatorFunction`, and a local instance of this class should be passed to approximation tool. The purpose of this change is avoiding the usage of static data and thus improving thread safety.

### Modeling Algorithms

- The methods `Value(...)`, which return distance between two extreme points, for all algorithms in `Extrema` package have been replaced by methods `SquareDistance(...)`, returning square distance between source and extremum points.

### Visualization

- The default behavior of `WNT_Window` class in part concerning the window background has changed. Its constructors set the window flag (`WDF_NOERASEBKGRND`) that suppresses the window background. To draw the background, this flag should be cleared explicitly.
- Behavior of two standard presentation builders from `MeshVS` package (`MeshVS_ElementalColorPrsBuilder` and `MeshVS_NodalColorPrsBuilder` classes) has been changed so that both could produce the same visual results when "color reflections" mode is ON (`MeshVS_DA_ColorReflection` drawer attribute controls this mode).

Material aspect is based on standard `PLASTIC` material, but with specific reflection properties used in both builders.

`PLASTIC` is most suitable for colored mesh elements as it is "non-physic" so the interior color given to `Graphic3d_AspectFillArea3d` constructor is then passed to OpenGL instead of the material colors, just scaled by the material reflection coefficients (ambient, diffuse, specular and emissive).

For example, in the `TelUpdateMaterial()` function in `OpenGL_attri.c.`, only two first coefficients are non-zero and equal to 0.5, to get the resulting color maximally similar to the input one (and, for instance, the one shown in the color scale). While `MeshVS_ElementalColorPrsBuilder` does not care about this scaling (it is performed automatically), `MeshVS_NodalColorPrsBuilder` now scales the input colors provided by an application in the same way as `TelUpdateMaterial()` function.

- The behavior of sensitive entities from `Select3D` package (`SensitiveWire`, `SensitiveSegment`, `SensitiveCurve`) has been modified. Verification whether the mouse point projection onto the segment's line falls inside the segment has been added. Two new methods: `SelectBasics_BasicTool::MatchSegment()` and `Select3D_SensitiveWire::GetEdges()` have been added to centralize detection of segments.

### Application Framework

- OCAF toolkit (TKLCAF) has become independent from OCCT modeling libraries. Now it collects all basic OCAF functionality useful for creation of arbitrary applications, while TKCAF toolkit contains attributes dealing with OCCT shapes and topological naming which are useful only for the applications based on OCCT geometry and topology.
  - Package `TNaming` has been moved from TKLCAF to TKCAF toolkit. Package `TFunction` has been moved in the opposite direction - from TKCAF to TKLCAF toolkit



- References to TNaming package are removed from TDocStd\_Document and TDocStd\_XLinkTool classes;
- Several OCAF attributes dealing with geometrical data that relied on TNaming attributes are extracted from TDataStd package to a new TDataXtd package (classes Position, Constraint, Placement, Geometry, Point, Axis, Plane, Pattern, PatternStd, Shape, Array10fTrsf and HArray10fTrsf; enumerations ConstraintEnum and GeometryEnum; package methods IDList() and Print()).

In connection with this change:

- User applications that rely on TNaming classes must be checked for links to TKCAF toolkit.
- User applications that implement their own XML or binary persistence and rely on TNaming classes must be checked for persistence drivers to be derived from corresponding standard drivers supporting TNaming classes (XmlDrivers or BinDrivers).
- User applications that use attributes from TDataStd package that have been moved to TDataXtd package (see list above) should be updated correspondingly; in addition environment variable CSF\_MIGRATION\_TYPES must be defined pointing to the file MigrationSheet.txt located in ros/src/StdResource directory. The latter is needed for correct reading of these attributes from OCAF documents stored by previous versions of OCCT. If environment variable CSF\_MIGRATION\_TYPES is not defined or is defined incorrectly, old attributes will be silently ignored on reading.

### Building Tools

- Since this release building Open CASCADE Technology with Visual C++ 6.0 compiler is not supported officially (though project files for VC++ 6 are still provided).
- Projects for building OCCT with Visual C++ 9.0 are provided.

### Bug Fixes



- Since last minor release (version 6.3) Open CASCADE 6.3.1 incorporates **80** modifications (bug fixes, enhancements and other corrections). For details, refer to [Appendix](#).



## Appendix: Open CASCADE 6.3.1 Modifications

- [Foundation Classes](#)
- [Modeling Algorithms](#)
- [Visualization](#)
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### Products

- [OMF](#)
- [Express Mesh](#)
- [DXF](#)
- [ACIS interface](#)

Foundation Classes, 7 modifications	
ID	Short Description
20212	Implement mechanism for investigation of memory problems
20501	Problem of the OCCT thermal units conversion
20515	OCC parallelization
20550	Miscellaneous OCC improvements, including improved thread-safety
20594	Bug in TColStd_PackedMapOfInteger regarding storage of negative values
20872	Correctives in some CDL files for package gp
20958	Suppression of duplicated templates
Modeling Algorithms, 34 modifications	
ID	Short Description
17194	Incorrect section of a face got by extrusion of an arc of circle
20904	BRepBuilderAPI_Sewing produces a shape with big tolerance
20201	Class BRepFeat_SplitShape raise exception if one of splitting wire tangent to boundary face.
20204	Improvements from Open CASCADE Forum for Modeling Algorithms
20222	Chamfer operation produces incorrect shape
20297	Boolean operations (for ex. fuse) fail for given shapes
20328	Integration of FIP "Improving thrusection algorithm"
20329	SALOME SMESH: Convert Quadratic and Group on GEOM
20373	Shading doesn't work on b-spline sphere
20391	Shape deforming via 3 scale factors raises error.
20413	Section is wrong for given shapes
20465	Wrong checkshape results for shells.
20524	Wrong intersection



20616	Failed to save/restore empty face
20627	Memory leaks in BRepMesh
20683	BRepExtrema_ExtCC produces incomplete result.
20766	Intersection of cone with plane in apex point crashes
20793	BRepOffsetAPI_MakeOffset produces incorrect result
20807	Construction of pipe among a helix containing several convolutions fails
20817	Improvement of Extrema performance
20823	BRepFilletAPI_MakeFillet2d fails to build a chamfer
20846	Few fixes and improvements for OCCT BRepMesh triangulator
20864	SALOME GEOM: Partition issue.
20866	BRepOffsetAPI_MakePipeShell fails on sections having different form
20959	Rubbish methods in BRep_Tool and unused data fields in BRep_CurveOnSurface
20964	Wrong result of cut operation for given shapes
21010	Method BSpl CLib::RemoveKnot returns invalid result for cases when multiplicity equal to degree
21014	Improvement of TopAbs and TopoDS packages
21121	Triangulation on face is not built
21122	Incremental mesh error
20127	Salome GEOM: Object from Cut operation non valid
21140	Elimination of warnings on 64 bit SunOS platform
21141	Elimination of compilation problem on 64 bit SunOS platform
<b>Visualization, 13 modifications</b>	
ID	Short Description
19820	3D discrete topology (voxels)
20205	Improvements from Open CASCADE Forum for Visualization
20311	Improvement of Select3D_SensitiveFace.cdl
20459	Diagnostic console output in Graphic3d_Group_2.cxx
20596	Eliminate background flicker in 3D view
20644	Color reflection mode distorts the result colors
20740	ChoosePixelFormat fails to find the correct number of Depth bits
20773	Incorrect interactive detection of curves and segments
20785	Surfaces are empty after shading
20802	V3d_View: Text size used by V3d_ZBUFFER'ed trihedron is too large
20847	NIS does not find Gl include files on SunOS 5.10
20957	Number of allocated V3d_VIEWS is limited to 100
21126	Infinite lines appear during selection in the local context on attached shape
<b>Application Framework, 6 modifications</b>	
ID	Short Description
20768	Dimension tolerances and materials can not be stored and retrieved in OCAF document.
20303	Incorrect writing of Stop shape Index in NamingDriver of Binary format
20324	Integration in OCCT modifications
20455	Unset string in case of input argument value is a null handle
21004	Naming mechanism improvement
21093	Separation of OCAF to Lite and Standard parts completion



Data Exchange, 4 modifications	
ID	Short Description
19936	Incorrect search for Representati onl tem in STEPControl_Writer
20200	Class ShapeAnalysis_FreeBounds can not make wire from edges with not shared vertices.
20707	Implement missing function XCAFDoc_DimTolTool::GetDatum
21124	Mapping between IGES entities and result shapes is lost on attached file.
WOK, 5 modifications	
ID	Short Description
13055	Extraction error in Transfer_ActorOfTransientProcess.cdl file
20122	Incorrect work of header extraction on Windows platform
20825	Errors during compilation for new version of WOK
20850	umake command does not work on SunOS10
21135	The BinPugin.vcproj is generated incorrectly for vc8_64
DRAW, 2 modifications	
ID	Short Description
20838	Some problem with visualization in DRAW on Mandriva2008
21046	XShow raises an exception
Development Environment, 1 modification	
ID	Short Description
20784	Problem of configure.in of make procedure
Building tools, 1 modification	
ID	Short Description
20832	Implementation able to work common sln projects for building OCCT as a whole
Documentation, 1 modification	
ID	Short Description
20503	OCCT 6.3.0 Doxygen documentation is broken for Graphic3d package

### Product Bug Fixes

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

OMF, 2 modifications	
ID	Short Description
20826	Incorrect result of Boolean operation on simple cubic-like meshes
20925	Create grid tests of OMF Boolean operations



Express Mesh, 1 modification	
ID	Short Description
20907	Failure meshing a face with several touching wires
DXF, 1 modification	
ID	Short Description
20811	Faces based on RectangularTrimmedSurface are incorrectly exported
ACIS Interface, 2 modifications	
ID	Short Description
20994	Some improvements for ACIS
21123	Repackage Draw stuff for DXF and SAT interfaces

