



## Open CASCADE 6.1 Minor Release

### Release Notes (Public Version)

#### Overview

Open CASCADE Technology 6.1 is a minor release, which includes new features, improvements and bug fixes, over version 5.2.



Version 6.1 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 6.1 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 6.1.





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## Highlights

- Support of new platforms, compilers, and build systems:
  - Possibility to build with nmake command on the Windows platform;
  - Support of Tcl 8.4;
  - Support for 64-bit SUN, DEC, Linux and Windows platforms;
  - Open CASCADE can be successfully compiled on Debian Woody, Debian Sarge, Fedora Core 3.0, Mandrake 10.1 and Mandriva 2006 Linux platforms;
  - Support gcc 4.01 and earlier on Linux platform;
- The structure of Open CASCADE Technology modules, toolkits and packages has been optimized for use in applications;
- The memory manager and the mechanism of system exceptions and signals handling has been improved;
- Visualization:
  - Colour scale implemented in Open CASCADE 3D Viewer;
  - More convenient use of texture mapped fonts on Windows;
  - Performance improvements in mesh visualization services (MeshVS package);
  - Visualization of 3D elements in MeshVS added;
  - Dump of a 3D viewer scene into vector graphics in PS, EPS and EMF (under MS Windows) formats has been implemented
  - Anisotropic (axial) scaling, has been implemented in OCCT 3D viewer;
- Documentation:
  - The reference documentation is now generated by the Doxygen tool and provided as a separate package in a more complete, convenient, and up-to-date form;
  - Draw Test Harness User Guide has been updated in part of Visualization, OCAF, Topology and Data Exchange;
  - Foundation Classes User Guide has been revised and updated;
  - Distribution of data through OCAF tree has been described in a new white paper
- Numerous improvements in modeling algorithms;
- OMF (Open CASCADE Mesh Framework) product has been improved in mesh operations, visualization and data exchange with NASTRAN;
- Parasolid up to schema 16 is read



## New Features

### Documentation

- Starting from this release the reference documentation is delivered in the form of HTML-based documentation generated by Doxygen tool.

### Foundation Classes

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

- The implementation of memory manager in Open CASCADE has been completely revised; now the optimized memory manager is implemented as a C++ class (`Standard_MMgrOpt`) and thus it is possible to use it in advanced applications requiring separate optimized memory heap. Note that the behavior and the interface of default memory manager remain the same: as previously, methods `Allocate()`, `Reallocate()` and `Free()` of the package `Standard` shall be used for memory allocation and de-allocation, and environment variables `MMGT_OPT` and others – for adjusting parameters of the default memory manager. See Foundation Classes documentation for details.

Besides, this improvement leads to a noticeable decrease of the size of Open CASCADE libraries on Linux and Windows platforms (from 5 to 10%).

- A new class `TColStd_PackedMapOfInteger` has been created. It can be used everywhere instead of the old class `TColStd_MapOfInteger` (also the new iterator type is defined). This improvement is important wherever the maps are expected to be great in size or if the performance of this map is important. The new class works from 2 to 30 times faster with common operations, such as add, remove and iterate.

The new class `TColStd_PackedMapOfInteger` inherits the user API of `TColStd_MapOfInteger`, however, there is no relation between these classes, they cannot be passed one for another or assigned one to another.

Two additional methods `GetMaximalMapped()` and `GetMinimalMapped()` have been created in the class `TColStd_PackedMapOfInteger`. These methods return the maximum and the minimum integer among all values contained in the Map. They use the internal architecture of this map and run much faster than the equivalent map iteration.





### Modeling Algorithms

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

### Visualization

- A color scale has been implemented in the OCC 3D Viewer on the basis of Visual 3d\_Layer class. The new class V3d\_ColorScale provides a possibility to show a color scale typically used in FEA applications right inside the 3d Viewer. This ready-to-use class brings numerous additional benefits to the user providing comprehensive capabilities to customize the color scale by manipulating the following parameters:
  - upper and lower values of data;
  - user-defined number of intervals (colors) in the color scale;
  - possibility to specify a list of colors explicitly (user-defined colors) to override default ones;
  - optional text labels for each interval (color) which would override default labels (numbers) generated automatically for each boundary between two intervals;
  - color scale caption;
  - width and height;
  - position (X and Y coordinates of the top left corner of the color scale in the 3D view window).
- A new exporter of graphic objects has been implemented in Open CASCADE Technology (Visual 3d\_View: :Export). This tool dumps objects displayed in 3D viewer into a vector graphics file in PS, EPS and EMF formats (under MS Windows), preserving such geometrical settings as display mode, colors, thickness of lines and others.

For correct presentation in shading and shrink modes the tool uses an algorithm similar to “Hidden lines removing”. The tool is able to save different colors or gradient fills of faces of an object in the PostScript document with these features, if this is the case.

An object extracted from 3D viewer is split into a set of primitives that can be stored in the PostScript document. OpenGL feedback buffer is parsed to obtain information about their properties.

The tool can sort primitives either using the BSP tree or by “simple sort”, comparing center point depths of primitives. The building of the BSP tree takes quite a time, but guarantees proper visualization of objects, while “simple sort” takes less time, but in some cases produces incorrect results.

The mechanism of feedback mode in OpenGL allows rendering the text using vector fonts and omitting line width, text strings and other properties.

Apart from PostScript format support, export to such platform-dependent format as EMF has been implemented. Since it is necessary to use the gradient fill feature, the version of Windows must be Win98, 2000 or later.

In its present state the tool imposes some limitations:

1. It supports PDF, PS, EPS and EMF (for Windows OS only!) formats. However, support for other formats can be introduced easily.



2. As the EMF format does not support transparent images, all markers in the documents become rectangular.
3. Open CASCADE Technology supports numerous interior styles, but currently the objects can be solid only. Note that line pattern is supported in produced documents.

The tool is used as follows:

- o Handle( V3d\_View ) aView = ...
  - o aView->View()->Export( "myfile.ps", Graphical3D\_EF\_PostScript );
- The **MeshVS** package now supports visualization and interactive selection of linear (first order) 3D mesh elements (for example, prism, pyramid and other). These elements can be shown as a set of polygons corresponding to faces. To use 3D elements, the application must provide the method DataSource::Get3DGeom, which returns a description of 3D element topology by MeshVS\_Array1OfSequenceOfInteger. The array index corresponds to the face number. The value describes a set of vertices, placed in a correct order, of which the face-polygon consists. In order to simplify the filling of this structure, class DataSource3D provides methods to fill an array with pyramid or prism topology. The sensitive entities for 3D elements are constructed automatically when the MeshVS\_ET\_Volume selection mode is computed using Get3DGeom.
  - The user can now control presentation of objects in the Viewer during their transformation. Using method SetTransformPersistance(), it becomes possible to forbid object rescaling, or to force an object to behave like static trihedron in the corner of the view scene, etc.

Obviously, the implementation of this functionality should take into account both visualization and selection of presentations. However, for this release 6.1 only the visualization part has been implemented, however, this improvement still remains very helpful in certain cases when it is necessary to display non-selectable objects with some degree of invariance with respect to view scene transformations.

Example:

- o Handle( PrsMgr\_PresentableObject ) anObj = ...
  - o anObj -> SetTransformPersistance( Graphical3D\_TMF\_ZoomPers );
- A new Draw command vtext has been added to implement the text alignment parameter (horizontal & vertical)

Implementation:

vtext string [x y z] HorizontalTextAlignment VerticalTextAlignment

For example: vtext Hello HTA\_CENTER VTA\_HALF

- A new callback mechanism has been added. Now it is possible to create user drawn presentations of interactive objects by using OpenGL commands.

The affected packages are: InterfaceGraphical, Graphical3D, OpenGL.

in InterfaceGraphical:

- new function is declared: call\_togl\_userdraw
- new types have been added:
  - CALL\_DEF\_BOUNDS - stores minimal and maximal values for a bounding box,
  - CALL\_DEF\_USERDRAW - stores user data of a presentable object.

in Graphical3D:



- new type has been added: CBounds (see CALL\_DEF\_BOUNDS). Minimal and maximal values for Graphi c3d\_Group are now implemented using CBounds;
- new type has been added: CUserDraw (see CALL\_DEF\_USERDRAW);
- new method has been added: Graphi c3d\_Group: : UserDraw() - creates an UserDraw primitive;
- new method has been added: Graphi c3d\_Graphi cDri ver: : UserDraw() - adds UserDraw data to a structure.

in OpenGL:

- new element type has been added: Tel Userdraw;
  - new method has been added: OpenGL\_Graphi cDri ver: : UserDraw() - adds UserDraw data to a structure;
  - new method has been added: GetCallb ackTabl e() (declared in OpenGL\_callback.h) - returns a user methods table pointer.
- A new visualization feature - anisotropic (axial) scaling, has been implemented in OCCT 3D viewer. It has become possible to scale a 3D view along each coordinate system axis independently. You can refer to the documentation on Vi sual 3d\_Vi ewOri entati on class and Axi al Scal e() and SetAxi al Scal e() methods of V3d\_Vi ew cl ass for more details.

New methods added in V3d package:

- V3d\_Vi ew: : SetAxi al Scal e(sx, sy, sz) - scales the view along each coordinate system axis according to scale factors
- V3d\_Vi ew: : Axi al Scal e(&sx, &sy, &sz) - returns defined scale factors
- V3d\_Vi ew: : Axi al Scal e(dx, dy, axi s) - overloaded member function scales the view along the given <axis> on the basis of the mouse pointer displacement <dx,dy>

New methods added In Vi sual 3d package:

- Vi sual 3d\_Vi ewOri entati on: : SetAxi al Scal e(sx, sy, sz) - sets axial scale factors of the view
- Vi sual 3d\_Vi ewOri entati on: : Axi al Scal e(&sx, &sy, &sz) - returns current values of the axial scale factors

Note that 3D presentations with "zoom persistence" flag turned on are also insensitive to anisotropic (axial) scaling.

### Application Framework

- **Lite OCAF component has been implemented**  
The OCAF component uses a lot of memory during work. Even a simple application, which uses OCAF as a tool for data storing, occupies a lot of memory and reduces performance. The solution of this problem is to create new simplified (Lite or basic) OCAF toolkits. Thus, new OCAF toolkits based on old packages, without presentation-dedicated packages have been created. Presentation-dedicated packages, such as TPrsStd, PPrsStd etc, use visualization packages that use a lot of memory. This modification will significantly decrease memory use by applications that don't need visualization.  
See also: [Appendix 2](#)
- XmlOcaf and XmlXCAF persistent documents are written using the XML syntax defined in the corresponding XML drivers (packages Xml Drivers and Xml M\*). The new resource unit Xml OcafResource contains the definition of this XML grammar using the W3C XML Schema. As



part of XML persistence support, these definitions can be used by end users in XML validators or editors, together with persistent XmlOcaf documents. On the other hand, these XSD definitions are not needed when these documents are written or loaded by applications using OCAF persistence drivers.

The main XML Schema definition file, XmlOcaf.xsd, declares the XML elements "document" and "label" that form the document tree structure. Particular data types (OCAF attributes) are declared in XmlOcaf\_T\*.xsd files, each using or inheriting an abstract type: either ocaf:AbstractAttribute or ocaf:AbstractXPathLabel or one of ocaf:Abstract\*Att. Any new (application specific) OCAF attribute can declare a new XML element type using or inheriting one of these available abstract types, by a simple addition of the corresponding XSD declaration to this resource directory.

You can consult the file XmlXcaf.xsd to learn the way how the new XmlOcaf-based persistence schema can inherit basic XSD definitions.

The location of XSD resource directory is written into the persistent document by the code of class XmlLDivers\_DocumentStorageDriver, as part of the attribute "xsi:schemaLocation" in the head of the created XML file. This location is found by the algorithm:

1. The environment variable "CSF\_XmlOcafResource" is checked; if it is not empty, then it is used as the location of XSD resource files.
2. Otherwise the environment variable "CASROOT" is checked; if it is not empty, the string "/src/XmlOcafResource" is appended forming the location of XSD resource files.

Otherwise the location of XSD resource files is not written to the XML document. It does not prevent successful storage and subsequent retrieval of this document by XmlOcaf-based persistence schemas.

**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.

**Draw Test Harness**

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

Use:

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

**-<Boolean Operation>**

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

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

**/<Test Options>**

- R (disable small edges (shrank range) test)
- F (disable faces verification test)
- T (disable tangent faces searching test)



- V (disable test possibility to merge vertices)
- E (disable test possibility to merge edges)
- I (disable self-interference test)
- P (disable shape type test)

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

**#<Additional Test Options>**

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

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

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

The 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\ocafbrowser 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
```

- Four new DRAW commands for working with units have been created:
  - "unitparsing" - for the analysis of expression of a given unit;
  - "unitsdico" - dictionary of supported units;
  - "unitconvtoSI" - convert value from given unit to SI;
  - "unitconvtoMDTV" - convert value from given unit to MDTV system.
  
- A new DRAW Test Harness command (vscale) has been added in ViewerTest package to demonstrate and test the axial scale feature in DRAW.





Products

**OMF**

- 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\_Model Commands;
- optional Boolean parameters for removing only free node and the nodes becoming free has been added to the command 'removenode' of the SMDSTest\_Model Commands;
- 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 SMDSTool\_s\_NASMeshReader class).

In addition, a simple tool for writing SMDS mesh to NASTRAN has been implemented (class SMDSTool\_s\_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.

**Express Mesh**

- The **Express Mesh** algorithm has been introduced. This new feature is not an open source part of Open CASCADE Technology, but a separate product with a commercial license.

The algorithm can be used as a simple quick meshing algorithm. The resulting mesh can be used for visualization instead of standard Open CASCADE Technology meshing algorithm (BRepMesh package), as well as in other applications satisfied by the patch-by-patch meshing solution.

Express Mesh will be available under the site development license (as most other licensed software components at [www.opencascade.org/support/products](http://www.opencascade.org/support/products)). For details please contact [marketing.contact@opencascade.com](mailto:marketing.contact@opencascade.com).

## Improvements

### Technical Documentation

- ❑ Draw test Harness User Guide has been updated in parts of Visualization, OCAF, Topology and Data Exchange.
- ❑ Foundation Classes User Guide has been revised and updated.

### Foundation Classes

- ❑ Compilation warnings reported when using NCollection templates have been eliminated on Linux.
- ❑ Memory allocation in classes AsciiString and ExtendedString in TCollection package has been corrected so that possibility to have memory corruption is excluded.
- ❑ Copy constructor of the Standard\_Transient class has been corrected to initialize field count by 0, thus ensuring a correct value of this field when classes inheriting Standard\_Transient are copied.
- ❑ Open CASCADE memory manager (its function Standard::Allocate()) has been fixed to work correctly with free lists in case when *MMGT\_OPT=1* and *MMGT\_CLEAR=0* (memory losses and incomplete copying in method Standard::ReAllocate() are avoided).
- ❑ Some boolean operations memory leaks have been fixed.
- ❑ The template classes in the NCollection package have been made compilable under gcc 3.4.
- ❑ Some problems of incorrect working with units in the Units package have been fixed.
- ❑ The compiler warning "C4535: calling \_set\_se\_translator() requires /EHa" raised when compiling OSD\_signal\_WNT.cxx under MS Visual Studio 7.1 has been eliminated. For more details, please, see the comments in the source file.

Treatment of Stack Overflow signal on Windows has been improved. Now the handler makes an attempt to recover the stack calling the RTL method `_resetstkoflw()`. It restores the guard page, so as the next time this condition occurs the signal is raised as well. If recover is impossible due to lack of memory in the reserved stack space, the C++ exception `OSD_Exception_STACK_OVERFLOW` is raised as usual, but with a description string "Unrecoverable STACK OVERFLOW" instead of the regular text "STACK OVERFLOW".

- ❑ Calls to C character classification routines (`isalnum`, `isdigit` etc.) have been corrected throughout Open CASCADE Technology code so as to ensure proper conversion of char to int (via unsigned char).
- ❑ Friend functions declared in CDL files using the 'C++ alias' syntax throughout Open CASCADE Technology code have been ensured to be exported onto Windows platform.
- ❑ The `OSD_Chronometer` class has been fixed to return the correct value of spent time on Linux.
- ❑ Vector collection is used instead of Sequence in the `NCollection_UBTreeFiller` class, with the relevant algorithm of randomization.



Method Add() from NCollection\_UBTree class has been slightly modified to avoid extra data copying. The additional gain of performance is about 10%.

### Modeling Data

- ❑ Class BRepAdaptor\_CompCurve has been optimized due a bug fix which eliminates the segmentation fault when the Edge() method is used.

### Modeling Algorithms

- ❑ The BRepAlgoAPI\_Section algorithm now does not fail in cases when unacceptable small edges are present in the shape but situated at a long distance from the section plane. Earlier such edges were obstacles to perform an operation regardless of how long the distance from them to the area of intersection is. This improvement allows to perform section for a valid part of the model, while section for an invalid part will still be impossible (according to Boolean Operations limitations).
- ❑ The API of the BRepPrimAPI\_MakePrism algorithm has been improved: an optional argument has been added to the constructor of this class to disable surface canonization. It is useful when we expect U-isolines of built surfaces going along original curves while V-isolines being parallel to the direction of extrusion.
- ❑ The low level internal algorithm responsible for searching a rough intersection between a curve and a surface and used in Boolean operations has been improved.
- ❑ The section part of Boolean operations has been improved: for some cases the section between shapes is tested to reject produced edges that are too small (with only one vertex completely covering the edges). Now the result of section is valid for further processing.
- ❑ Intersection of a surface of linear extrusion with a plane parallel to the direction of extrusion now stably produces a direct line instead of a set of overlapped B-spline curves.
- ❑ Some errors in internal algorithms for the 'sprops' (surface properties) computing have been fixed.
- ❑ Treatment of 3d offsets on "lofts" (shapes constructed by the ThruSections algorithm) has been improved.
- ❑ A bug in the algorithm of BRepFill\_CompatibleWires, through which the resulting shape that should have been non-twisted became twisted, has been fixed.
- ❑ 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).
- ❑ The dependence of toolkits TKOffset and TKShHealing on TKDraw toolkit in debug mode has been eliminated.
- ❑ Class GProp\_GProps has been corrected to compute the length of b-spline curves taking into account continuity intervals (as class GCPnts\_AbscissaPoint does).
- ❑ Now the Offset Shape algorithm processes correctly the shapes based on pseudo-cylinders and pseudo-cones (surfaces that look like cylinders and cones but are in fact surfaces of revolution or BSpline-surfaces).
- ❑ The algorithm ShapeAnalysis\_FreeBounds has been optimized for performance by avoiding the quadratic dependence of the computation time on the number of free boundaries.
- ❑ The algorithm BRepAlgo\_Sewing has been improved so as to generate the same stable result repeatedly on the same input data.



- ❑ Memory leak (mentioned in thread 9220 on the OCCT Forum) in the code of the class `IntCurvesFace_ShapeIntersector` has been eliminated. Now the destructor of this class properly frees all allocated memory.
- ❑ Some algorithms responsible for wire analysis used in a face building process have been improved.
- ❑ The performance as well as the quality of intersection part of Boolean operation algorithm has been improved. The improvements mostly take place for shapes with BSpline surfaces and BSpline curves. The searching of intersection points is also improved for large BSpline surfaces and for the case when a number of intersection points exist between an edge and a face based on BSpline geometry. The maximum performance improvement according to the tests is up to 15 times.

### Visualization

- ❑ Several cases of uninitialized fields in Visualization and OCAF classes have been detected and corrected, to avoid problems when Open CASCADE Technology applications run with environment variable `MMGT_CLEAR` set to 0.
- ❑ Selection mechanism has been improved in order to clear the current selection after selecting an empty space with a rectangle when the local context is open.
- ❑ Visualization algorithm has been modified in order to allow simultaneous display of Bitmap fonts, Texture Mapped fonts and Textures.
- ❑ The `MeshVS_Mesh::Compute` method is now up to 20% faster than before. Earlier, in order to indicate which elements or nodes must be shown, this method filled the map of integer to Boolean with values, but since the quantity of nodes and elements is already known, now we use the array instead of the map.  
To make the `Compute` method faster its interface has been changed. Now the successors of `MeshVS_PrsBuilder` use `Array10fIntegerBoolean` instead of `MeshVS_Map0fIntegerBoolean`. If the application searches for an array element, it can use binary search because the array passed to `PrsBuilder` is sorted by integer key. To iterate through the array, it can use indices from `Array.Lower()` to `Array.Upper()`.
- ❑ Now it is possible to switch interpolation of texture color with ease. Just call the method `AI_S_TexturedShape::EnableModulate()` and the colors will interact with the light. Call `AI_S_TexturedShape::DisableModulate()` in order to forbid the change of texture colors according to the light.
- ❑ Earlier, in case when:
  - method `TriAngleSet` was used with array `Graphi c3d_VertexC`; and when
  - an array of polygons with colors assigned to its vertices was added to `Graphi c3d_Group`, the resulting shapes were non-transparent shapes even if the material of the fill area aspect was set to transparent. Now, this bug has been fixed and transparency is displayed properly in both cases.
- ❑ Thanks to the help of Mr. Sangsu Lee and Mr. Huques (OCC users), memory consumption by the application during the processing of a Z-buffered trihedron in a `V3d_View` object was greatly reduced through some changes implemented in the algorithm.
- ❑ A problem connected with internal iterator in the class `AI_S_Selection` after the removal of the current interactive object from the context was fixed.
- ❑ `StdPrs_ToolRFace::Next()` has been improved to avoid unnecessary recursion.
- ❑ `Graphi c2d_Primitive.cdl` has been corrected for efficient compilation on Gcc3.4.
- ❑ In `TPrsStd_AISPresentation::UnsetMode()`, method `AI_S_InteractiveContext::UnsetDisplayMode()` is now used instead of `AI_S_InteractiveContext::SetDisplayMode()`.



- ❑ Method `FindSelectedOwnerFromShape()` has been optimized.
- ❑ Now `MeshVS_Mesh` allows to draw mesh with no nodes or no elements without a crash.
- ❑ A fix on class `Graphic3d_TextureRoot` has been provided by OCC community member Francois Lauzon. This fix allows to create textures dynamically at runtime instead of loading them from a file.
- ❑ A new callback mechanism allows the user to create object presentation using OpenGL commands. To do so it is necessary to create an AIS part (`AIS_InteractiveObject`) and a driver part (Set of methods to handle a `UserDraw` presentation: `Pick`, `Display`, `Add`, `Delete`... These methods must be added to the callback table).  
The driver package must be compiled in a library and must be loaded instead of `TKOpenGL` (via `CSF_Graphic3d_Shr` variable).
- ❑ A problem with initialization of the OCCT graphic system has been found on WinXP 64-bit platform (CPU AMD Athlon 64). The exception `Aspect_GraphicDeviceDefinitionError` occurred and the message string "Access to station is denied" appeared whenever an OCCT-based application with OCCT libraries compiled for Win32 platform was launched on a WinXP 64-bit station. This problem has been successfully recognized and eliminated.
- ❑ **MeshVS package**
  - Mesh elements can be displayed using interpolated colors. In this case the colors are assigned to the nodes of mesh elements, not themselves, and the resulting color of each element is a gradient fill between its nodes. Previously, when a presentation was built, the visibility of nodes was analyzed instead of the visibility of elements. However, it is possible that an element is hidden but all its nodes are visible, as a result, the hidden element was displayed on the screen. Now this incorrect behavior has been fixed, and the visibility of mesh elements is taken into account instead of the visibility of nodes.
  - Mesh borders of faces were not visible in the shrink display mode during the visualization. Now this problem in MeshVS package has been fixed (the borders are colored with black).
  - It is possible to assign vector values to the elements or nodes during a mesh presentation. In this case a vector presentation is displayed. It looks like a mesh with vectors issuing from nodes or elements. In OCC5.2.2 this presentation was successfully built with default parameters. In OCC5.2.3 it did not work, because one parameter was not initialized. As a result already written applications could be broken. Now this problem has been fixed.

### Application Framework

- ❑ Binary persistence on Linux has been fixed.
- ❑ Xml persistence: ability to read and write the "reference section" of XmlOcaf documents has been added.
- ❑ Memory leak has been eliminated when generating messages during persistence storage/retrieval operations.
- ❑ Drivers for persistence (OCAF, Xml, and Binary formats) have been implemented for OCAF attribute `TDataStd_ExtStringArray`.
- ❑ `LDOMParser` now correctly treats the DOCTYPE declaration with external ID, i.e. containing the URI of a DTD file. As before, this DOCTYPE declaration is ignored in the parsed document and cannot be passed to the XML output by a subsequent call to `LDOM_XmlWriter`.
- ❑ In the constructor of `LDOM_OSSStream` class, call of the `Init(streambuf*)` method of the base class has been added. This prevents incorrect initialization of base class fields on some platforms.



### Data Exchange

- ❑ Stl API\_Reader (translator) has been corrected to apply the sewing algorithm on all faces at once (all faces are collected in one compound, then the sewing algorithm is applied) instead of sewing faces one by one.
- ❑ The retrieving algorithm of gp\_Trnsf in Xml format has been corrected.
- ❑ Writing of surfaces of linear extrusion to IGES files has been improved.

### Test Harness Draw

- ❑ The problem with presentation of some faces built on periodical surfaces in the DRAW application has been fixed.
- ❑ Command `igesbrep` in the previous version of DRAW always read all entities from an IGES file (visible and invisible) independently of the specified mode (whether it is ALL ROOTS or ONLY VISIBLE). Now this bug is corrected and the command reads entities in DRAW according to the mode.
- ❑ 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`
- ❑ The dependence of OCAF DRAW toolkit TKDCAF on high-level modeling algorithms (TKBOP, TKBool, TKFill et, TKPrim) has been eliminated. As a result, projects that need OCAF but do not need Boolean operations will not need to drag these toolkits.

### WOK

- ❑ WOK has been corrected in order to fix the problem with the `xcpp.template wok` compilation step on Windows platform.
- ❑ A command line for bison parser has been updated to avoid some compilation problems on Linux. Actually the old version of bison created output files with extension `*.tab.c` regardless of the input file extension, while the modern version of bison added the extension of the input file to the extension of the output file. As a result, the generated file was not seen by WOK. Direct definition of the output file name with the `-o` option resolved the problem.

### Samples

- ❑ **Visualization Performance Monitor** sample:
  - Some functionality has been implemented that was declared in sample dialogs but actually didn't work;
  - "Stop automatic testing" functionality has been added;
  - Enhanced sample look'n'feel (dialogues);
  - Some minor bugs have been fixed.
- ❑ A possibility to save and open files in binary and xml formats has been added to the MFC OCAF sample.
- ❑ Export to PostScript functionality has been added to Qt Import-Export and Hidden Line Removal (HLR) mfc Sample.
- ❑ A possibility of anisotropic scaling has been added in the Viewer3D mfc Sample.



## Building Tools

- ❑ The procedure of compilation of user applications built on the basis of OCCT binary distribution package has been simplified:
  1. The specification of `--with-tcl`, `--with-tk` arguments of the configure script are no longer mandatory. The configure script will search for `tcl/Config.sh`/`tk/Config.sh` files in default locations, such as `/usr/lib`, `/usr/local/lib` for Linux, `/opt/sfw/lib`, `/usr/tcltk/lib/itc` for Sun Solaris. Even if location of `tcl/tk` installation isn't found by configure script, it will continue working.
  2. In order to exclude the possibility of collisions between the `$CASROOT/inc/config.h` file supplied with binary distribution and the `$CASROOT/config.h` file generated by configure script, a step of creation of a symbolic link between `$CASROOT/inc/config.h` and `$CASROOT/config.h` has been added to the configure script, so if the user executes configuration the `$CASROOT/inc/config.h` file supplied with binary distribution will be replaced by the generated one.
- ❑ A possibility to build OCCT modules and the whole OCCT with the `nmake` command on Windows platform has been implemented.
- ❑ When building Open CASCADE Technology libraries from sources on Windows NT platform, macro `USE_OLD_STREAMS` can be defined to enforce it using the old-style streams library rather than new (STL) streams (used by default).

## Products

### Express Mesh

- ❑ The Express Mesh algorithm has been improved to avoid hanging and excessive memory consumption when meshing some narrow long faces.
- ❑ The objects of class `QMDData_Curve2d` dynamically allocated and stored in the field `myPCurves` of class `QMDData_Edge` were not deallocated if the method `RemovePCurve` was called, which lead to a significant memory leak on meshing operation. Now this memory leak in Express Mesh algorithm is eliminated.

### OMF

- ❑ A new parameter `isTopo` has been added to the `SMDSControl_BoundaryEdges` class. This parameter manages the way the edges are considered by the algorithm. The previous behaviour is preserved with the default value (`False`). If `isTopo` is `True`, every edge is considered as a part of the boundary, if it has different numbers of occurrences on mesh faces for (+) orientation and for (-) orientation. In this mode an edge between two faces with topologically opposite orientations is considered as a boundary. The topological boundary of a mesh can be used to check if the mesh is well defined with respect to orientation of its faces. This improvement adds a dependence of `SMDSControl` to the package `SMDSTools`.
- ❑ 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.
- ❑ Application crash when deleting an interactive object in the OMF Sample has been fixed.
- ❑ SMDSTool s\_STLReader class is now capable to store in the mesh faces the normals that are read from an STL file. This feature is optional. The default behavior has not been changed.
- ❑ Several improvements including performance improvements have been implemented and additional algorithms have been added to the Open CASCADE Mesh Framework.
- ❑ The QMchecktopo DRAW command has been corrected in order to output information messages only if errors are detected.

### DXF

- ❑ Translation of "Insert" entities with scales has been corrected.



## 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.

The OCAF component has been separated in two parts: Lite OCAF and complete OCAF. The Lite OCAF does not depend on OCC Visualization, which allows using OCAF in applications that do not use OCC visualization without need to link with big visualization libraries.

Please, see [Appendix 2](#) for detailed information.

### Supported Platforms

- Open CASCADE Technology now can be successfully compiled on Debian Woody, Debian Sarge, Fedora Core 3.0, Mandrake 10.1 and Mandriva 2006 Linux platforms:
- Open CASCADE Technology has been ported, compiled and tested on the following 64-bit platforms:

PLATFORM	COMPILER	CPU
Sun: Solaris 2.8	Forte 6.2	Ultra Sparc-5-10
Linux: Fedora Core 4.0	gcc 4.0	AMD64
MS Windows XP	VC 14.00.40310 for AMD64	AMD64

- Open CASCADE Technology has been ported to and compiled on Mandriva 2006 platform.





## Documentation

- Starting from version 6.1 Open CASCADE Technology has changed its approach to documentation. Now documentation is generated by **DOXYGEN**, an open source documentation system. Documentation is generated by OCCT modules (OCAF, Foundation Classes, Visualization, etc.).

Doxygen documentation is much more convenient to the users since it contains all OCCT Class Hierarchy in text and graphic modes, description of classes, methods, functions, enumerations, variables and typedefs.

Inheritance relations between classes are represented by means of graphs.

Class Browser is no longer provided.

## Foundation Classes

- The mechanism of system exceptions (signals) handling has been improved:
  1. All floating point errors are now converted to OCCT exceptions inheriting the `Standard_NumericError` class.
  2. The function `OSD::SetSignal()` has been enhanced on Windows platform. Now it can work using either the `SetUnhandledExceptionFilter` Win32 API call or the `_set_se_translator` MS Visual C++ runtime library call. The needed method is chosen using the `OSD::UseSETranslator` function.

A large number of floating point errors in various modeling algorithms (extrema, boolean operations, draft features, surface intersector, local and global properties of shapes, shape healing) have arisen after this improvement. They were all fixed, and the reliability and stability of OCCT code in general became much better.

Please take special precautions when migrating to the version 6.1 if you used some special handlers for numeric exceptions in your code.

- 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.
- Now it is not possible to create `ExprIntprp_GenExp`, `ExprIntprp_GenFct` or `ExprIntprp_GenRel` explicitly, static methods `Create` in each class must be used instead.
- Currently most of macro from package `NCollection` are incarnated using the corresponding templates. There is no reason to use such macro in future – use of templates is recommended.
- Due to the reimplementation of the memory manager as described above, a number of obsolete and internal functions and global variables that were previously exported from `Standard` package are now eliminated. Among them are:
  - method `Standard::Free()` with two arguments (the second one was useless)
  - methods `Standard::Allocate()`, `MFree()`, `MReallocate()`, `RReallocate()` – only the methods `Allocate()`, `Free()` and `Reallocate()` will be used
  - methods `Standard::About()`, `StorageManagerInit()`, `(Re)AllocatedLength()`, `StorageManagerDump()`, `Display()`
  - global variables `MMgt_Opt`, `MMgt_Clear`, `theFreeList`, `theFreeListSize`

Applications that might refer to these objects will have to be corrected.





- ❑ Global functions `HashCode` and `IsEqual` for `TCollection_AsciiString` class are now defined in `TCollection_AsciiString.hxx` file rather than as 'C++ alias: friend' in the CDL file. This affects a scope of visibility of these functions according to the C++ standard (relevant to the MSVC++ 7.1 compiler only).
- ❑ The class `NCollection_UBTreeFiller` has been changed in one aspect: the parameter `isFullRandom` has been added in its constructor. By default it is `True`, which ensures the correct randomization of large tree sizes (50,000 or more). If this parameter is `False`, the randomization proceeds in a window `RAND_MAX` wide (32767 values).
- ❑ The name of SI unit for minutes has been corrected in `Units.dat`: **min** instead of `mn`. Thus now 'min' will be recognized as a minute, not as milli-inch as previously. Note that unit 'mn' still can be used as an alias to minute.

**Modeling Algorithms**

- ❑ The history of Prism & Revolution algorithm has been redesigned to take into account use-cases when the resulting shape is an open shell.

**Visualization**

- ❑ Internal `TEL_COLOUR` structure declared in `InterfaceGraphic_tel.h` file and widely used in Open CASCADE Technology visualization engine has been changed to include alpha value, in addition to three color components.
- ❑ The performance bottleneck has been fixed by using `Select3D_SensitiveGroup::Add` method with a list as an argument. The old version used method `Add()` with one sensitive entity as an argument. This variant of method works slowly because every time it runs through the list looking for an entity. Now the user can create a full list and pass it to `Select3D_SensitiveGroup`.
- ❑ `MeshVS_Mesh::ComputeSelection`: Previously the default selection mode was 0. Now this mode is intended for selection of the whole mesh. The default selection must be of any value different from `MeshVS_EntityType` flags and zero.
- ❑ The behavior of the "meshsel mode" `DRAW` command has changed as follows:
  - 1) if selection mode = -1, the command closes all local contexts, therefore selection is impossible;
  - 2) if selection mode = 0, only 0 selection mode (whole mesh) is possible;
  - 3) if selection mode > 0, the 0 selection mode becomes inactive (this solution fixes the bug with slow selection).
- ❑ Some class names have been changed. The following table describes the correspondence between old names of classes instantiated in the MeshVS package of the previous versions and new names of the current version (since 5.2.1):

Previous version (5.2)	Current version (6.1)
<code>MapOfColor</code>	<code>DataMapOfIntegerColor</code>
<code>MapOfMaterial</code>	<code>DataMapOfIntegerMaterial</code>
<code>MapOfInteger</code>	<code>DataMapOfIntegerBoolean</code>
<code>MapOfOwner</code>	<code>DataMapOfIntegerMeshOwner</code>
<code>MapOfText</code>	<code>DataMapOfIntegerAsciiString</code>
<code>MapOfTwoColors</code>	<code>DataMapOfIntegerTwoColors</code>
<code>MapOfVector</code>	<code>DataMapOfIntegerVector</code>
<code>TwoColorsToIdsMap</code>	<code>DataMapOfTwoColorsMapOfInteger</code>
<code>ColorToIdsMap</code>	<code>DataMapOfColorMapOfInteger</code>



The names of iterators have been changed in accordance with Open CASCADE Technology rules, for example:

For `DataMapOfIntegerVector` the iterator now has the following name: `DataMapIteratorOfDataMapOfIntegerVector` and etc.

Other changes:

1. Methods in the `MeshVS_Drawer` now take the `MeshVS_DrawerAttribute` enumeration, not the `Integer` one as they did earlier (however this approach has a serious drawback, bug 7088).
  2. In the constructor of `MeshVS_PrsBuilder` and its successors the first parameter is `Handle` to `MeshVS_Mesh`, but the builder contains a pointer to `MeshVS_Mesh` as earlier and in constructor it extracts pointer from handle.
  3. The new `MeshVS_DrawerAttribute` enumeration item has been added: `DA_ComputeSelectionTime`, if its value is true, a string with information about the run time of the `MeshVS_Mesh::ComputeSelection` method will be passed the standard stream of output.
- Currently Texture Mapped fonts are used by default. In order to switch the default font type it is possible to use `TexFontEnable` or `TexFontDisable` static methods from the `Graphic3d_AspectText3d` class. To switch texture mapped fonts on/off the `Graphic3d_AspectText3d::SetTextureMappedFont(Standard_Boolean)` method should be used with the `Standard_True/Standard_False` argument. First, a `Graphic3d_AspectText3d` object should be created and initialized and then this object can be used in a usual manner, i.e. passed to the `Graphic3d_Group::SetPrimitivesAspect()` method. Without explicit calls to `Graphic3d_AspectText3d::SetTexturedMappedFont()` the objects get created with the use of texture mapped fonts (if `TexFontDisable` was not called previously).
  - A certain regression in the implementation of transparency was registered in the previous version of OpenCASCADE Technology. This problem was eliminated, however, the transparency problem in general partially remains even for a single transparent object. Certain undesirable visual effects can be still seen. They are caused by lighting calculations performed by OpenGL: the calculated color of two triangles A and B is different (one lying on a part of the shape more distant from the eye, another on a closer part, both oriented differently with respect to light direction). Therefore, the result of blending "background color -> 0.9 \* A color -> 0.9 \* B color" is different from blending "background color -> 0.9 \* B color -> 0.9 \* A color". In other words, the resulting image still depends on the order of triangles in the object triangulation. We hope to improve the situation in future releases.
  - If the interior and back interior colors are different, then `MeshVS_Mesh` draws the back and the front sides of a face using correct colors.
  - A mistake was corrected in `Graphic3d_AspectText3d::Values()` method, now it sets `<ATextureMappedFont>` output argument correctly.

### Data Exchange

- Full circles are now translated into a circle IGES entity instead of NURB entity in the past.
- A modification has been made to avoid setting the DONE status for a valid shape after `ShapeFix_Shape`.





### Porting

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

### CAD Models

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

### Building Tools

- ❑ TKj cas toolkit has been moved from Foundation Classes module to a separate module Wrappers.

### Compilation

- ❑ **Warnings are now produced in case of a wrong compilation environment (e.g. missing WNT macro);**

If compilation is performed by the Visual Studio compiler but "WNT" option is not defined, the pre-processor returns the following error: "Wrong compiler options have been detected. Add /DWNT option for proper compilation!!!!!"

For UNIX platforms if neither HAVE\_I OSTREAM nor HAVE\_I OSTREAM\_H is defined in config.h file or in compiler options, the pre-processor returns error: "check config.h file or compilation options: either HAVE\_I OSTREAM or HAVE\_I OSTREAM\_H should be defined".

The pre-processor returns a similar error if neither HAVE\_LI MI TS nor HAVE\_LI MI TS\_H are defined.

### Binaries on Windows

- ❑ Starting from Open CASCADE Technology 6.1 binaries on Windows are delivered compiled with MSVS 7.1 (.NET).

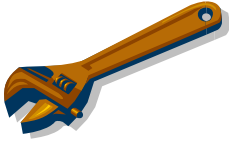
### 3<sup>rd</sup> Party Products

- ❑ Starting from Open CASCADE Technology 6.1 the platform is delivered with **QT 3.3.3** on Windows instead of previously used version of QT 2.3.0. This is caused by the fact that binaries on Windows are delivered compiled with MSVS 7.1 (.NET).
- ❑ Starting from Open CASCADE Technology 6.1 the platform is delivered with **Tcl 8.4** instead of previously used version 8.3.
- ❑ Starting from this version 6.1 Open CASCADE Technology is delivered with **fltk 1.1.6**.





## Bug Fixes



- Since last public release (version 5.2) Open CASCADE 6.1 incorporates more than **260** modifications (bug fixes, enhancements and other corrections) over version 5.2. For details, refer to [Appendix 1](#).



## Appendix 1: Open CASCADE 6.1 Bug Fixes

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

### Products

- [OMF](#)
- [Express Mesh](#)
- [Surfaces from Scattered Points](#)
- [DXF](#)
- [Canonical Recognition](#)
- [Parasolid](#)

Foundation Classes, 36 bug fixes	
ID	Short Description
984	LDOM hangs-up attempting to read a file containing a reference to a .dtd-file.
5421	Support of STL streams should not be mandatory on WNT
6143	Corrected handling of exceptions on WinNT.
6184	Compilation warnings when using NCollection templates.
6583	Malloc/free problem in NCollection_Vector.
6650	Problems with SALOME compilation on CASCADE 5.2 installed from CD to Linux RH8.
6794	AsciiString corrupts memory if MMGT_OPT is equal to 0.
6804	Class Standard_Transient has copy constructor which does not initialize field count.
6865	Standard::Allocate works incorrectly if MMGT_OPT=1 and MMGT_CLEAR=0
7034	NCollection List and Sequence modified in SD project
7253	Compilation at g++ (GCC) 3.3.4 (Debian 1:3.3.4-13)
7287	Problem of Memory Leak
7349	New algorithm for binary tree of bounding boxes
7639	NCollection_Vector works incorrectly with rare data
7698	Wrong behaviour of NCollection_Vector::operator=
7757	Providing possibility to compile OCCT on DEC platform
8345	Bugs in ExprIntrp and Expr packages (reported by FKL)
8500	The spent time received from OSD_Chronometer has a wrong value for OS LINUX
8503	Hide the objects to deny unauthorized operations with memory manager
8555	Results of FIP "Provide warnings in case of wrong compilation environment"
8595	Accessing freed memory in NCollection_UBTreeFiller
8682	General revision of OCCT packaging
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



9719	extracted operators from cdl are not exported
9770	Incorrect work of ExprIntrp package
9795	Definitions of the global functions HashCode and IsEqual are defective for compiler version 7.0
9827	Ensure passing unsigned char to character classification routines (e.g. isdigit())
9848	Specifying an invalid name of unit produces exception.(for example tonne instead ton)
9850	Parsing of expression containing parentheses is incorrect.
9866	Eliminate compile warning C4535: calling _set_se_translator() requires /EHa
10415	Some NCollection template classes are not compilable with gcc3.4.1
11403	NCollection_UBTreeFiller is too slow
11404	TColStd_MapOfInteger enhanced
11568	UnitsAPI package uses incorrect symbol for minutes: mn instead of min
<b>Modeling Data, 4 bug fixes</b>	
ID	Short Description
5696	Exception in BRepAdaptor_CompCurve due to uninitialised variable CurIndex.
6503	Incorrect calculation of bounding box for infinite surfaces
10603	Fuse of torus fails
10604	Fuse fails
<b>Modeling Algorithms, 74 bug fixes</b>	
ID	Short Description
569	Can not intersect trimmed plane and cone surfaces.
593	Projection of a 3D point on surface using GeomAPI_ProjectPointOnSurf works incorrectly
5157	DRAW commands vprops and sprops with tolerance 1.e-6 huge on attached shape.
5806	BRepOffsetAPI_MakeThickSolid fails to build a result
5886	An optional argument is added in the BRepPrimAPI_MakePrism class constructor to disable the simplification of linear extrusion surfaces to planes.
6063	Exception is raised during command "sprops" with epsilon <1 on attached shape in DRAW.
6181	Incorrect result of boolean cut
6182	Incorrect result of boolean cut
6272	Incorrect results of gluing operation.
6277	Cut operation does not produce any result.
6278	Fuse operation produces incorrect result
6279	Incorrect section between two solids
6281	Cut operation is invalid (caused by wrong section between arguments, it contains a new edge instead of the edge from one of the arguments.).
6282	Cut operation is invalid (caused by wrong section between arguments, it contains edges with one vertex, but these edges are neither degenerated not based on closed in 3D or 2D curve.
6289	Internal error (code 101) in Section computation.
6334	Invalid result of MakeCylindricalHole when input solid is reversed.
6371	Performance of some classes improved.
6390	ExpressMesh algorithm has been integrated as a separate product.
6450	Bcommon operation does not work with operand halfspaces made of two parallel infinite planar faces, and having a common space between these faces.
6474	Intersection of a plane and an extrusion surface gives incorrect result (4 overlapping bspline curves instead of 1 direct line).



6502	Wrong calculation of ShrunkRange algorithm.
6538	Empty result obtained by operation "common" for couple of faces
6554	Intersection shell-plane gives a disconnected set of edges
6581	Cases check 002 F2-F5, H2-H5 is not finished. Loop on OFFSETSHAPE for psphere.
6602	New Boolean operations return error on Solaris 2.8 with Forte 6.2
6605	Exception at offset operation on Solaris 2.8 with Forte 6.2
6615	Some sources are modified to introduce a special algorithm of partition.
6766	Invalid result of intersection of a bspline face with a plane
6811	Unstable work of the checkshape on the attached file.
6841	Invalid shape produced by Boolean common
6841	Invalid shape produced by Boolean common
6842	Stack overflow in Gluing operation
6862	Tool for checking arguments of Boolean Operations
6891	Triangle algorithm exits the application
7166	Improvement of API of ThruSection
7189	Analysis of possible regressions in Boolean operations
7354	Bad result of thrusections operation.
7372	Invalid conversion 2D periodic BSpline curve to Bezier segments.
7582	Bad result of offset operation
7626	Bad result of bcut operation
7668	BRepOffsetAPI_MakeOffset wrong generated list
7723	Operator precedence in Correct_Wire
7966	Work of BRepClass3d_Intersection3d contradicts documentation
8169	Wrong Geom2dAPI_InterCurveCurve segment
8228	Exception is raised during the vprops in DRAW on the attached shape
8259	FindColor method in Aspect_ColorScale
8372	Twisted result of ThruSections operation
8445	Bug in Prism & Revolution history
8498	Offset operation crashes the application
8543	Summary of PAL 8284, PAL8293
8545	History of Offset algo is incorrect
8559	Integration of several patches made for SD projects
8561	Set of bugs in OCC Algos linked with Gluing operation
8712	gcc3.4 porting
8797	Invalid BSpline curve length computed by means of GProp_GProps
8842	Errors in Offset Shape algorithm working with bspline and surface of revolution
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
9936	Fix on boolean operations for Salome project.
10160	BOP performance improvement.
10232	Exception "Segmentation violation" in Boolean Operations.
10435	error in fillet after Cut of torus from box.
10525	Integration of ShapeHealing improvements for DataKit CATIA V5 connector.
10605	Fuse fails.
10606	Fuse fails.
10842	Bug in BOP.





10846	Cut operation produces invalid result.
10886	Memory leak in IntCurvesFace_ShapeIntersector.
11457	The method Edge of BRepLib_MakePolygon works incorrectly on a closed wire
11493	The OCC bug is initialized by SALOME bug IPAL11259.
11677	The OCC improvements that are initialised by SALOME bug PAL11298
<b>Visualization, 57 bug fixes</b>	
ID	Short Description
978	Transparency problem in AIS
5682	AIS_InteractiveContext::Remove does not remove the object from memory.
5988	A textured shape (AIS_TextureShape) becomes black if we display an AIS_PlaneTrihedron with it.
6145	Incorrect rendering of a transparent object
6247	Color scale in OCC 3D Viewer.
6268	Incorrect functioning of transparency
6496	Performance problem of loading big STL shape by meshfromstl DRAW command.
6562	No default interior colour and interior style in MeshVS_Mesh.
6571	Bitmap and Textured mapped fonts optimization.
6571	Bitmap and Texture mapped fonts optimization
6652	Problem of mesh selection.
6802	Exceptions in visualization when MGMT_OPT=0 and MGMT_CLEAR=0
6802	Exceptions in visualisation when MGMT_OPT=0 and MGMT_CLEAR=0
7014	Now user defined markers are drawn
7068	Incorrect behaviour of a method in AIS_InteractiveContext.
7088	MeshVS_Drawer methods should accept integer values instead of enumeration, to ensure customisability
7089	Sometimes the text is not displayed when using texture mapped fonts.
7116	High-level control of view transformations of an object.
7186	Selecting an empty space with rectangle doesn't clear the current selection, when there is an opened local context
7190	Texture-mapped fonts don't work, as soon as any old-style (bitmap) text is displayed
7251	Provide some means to control selection activation on displaying an object in global context
7288	Texture with text
7456	Text alignment attributes
7667	Export contents of view to a vector graphic file
7726	Memory leak with the trihedron display
7781	Mistake in TPrsStd_AISPresentation::UnsetMode()
7824	Transparency doesn't work for Graphic3d_VertexC
7833	Transparency doesn't work in the case of array of polygons with colours assigned to vertices
7901	Visualization of 3D elements in MeshVS
7902	Acceleration of MeshVS_Mesh::Compute method.
8306	AIS_Selection becomes invalid after deletion of a selected object
8525	Regression: update of viewer has low performance on Linux
8563	Incorrect comparison in StdPrs_ToolShadedShape::Normal
8565	Colour Scale are not updated after change of text height
8568	Regression of transparency implementation as compared to OCC 3.1
8623	An additional method to manipulate the behaviour of texture colors at light.



8722	Optimization of StdPrs_ToolRFace::Next() method
8757	gcc3.4 returns error at Graphic2d_Primitive.hxx compilation
8854	Regression in primitive arrays
9026	[OCC Forum: thread 8263] Missing braces in FindSelectedOwnerFromShape function
9127	Visualization problems on Mandrake 10 platform
9489	Very low performance of MeshVS presentation on preselection
9517	Wrong use of method glTexCoordPointer() in OpenGL_PrimitiveArray.c
9657	AIS_InteractiveObject can't be selected
9658	Crash on mesh with no nodes or no elements
9659	Interior and back interior color are the same
9766	Add OwnerDraw primitive
10059	Possibility to initialize texture by dynamically created image
10381	Xdps_CMPLRS.edl does not add path to X11_INCLUDE for Linux platform.
10528	Cannot initialize graphic device on WinXP 64-bit.
10702	MeshVS_NodalColorPrsBuilder does not take into account visibility of mesh elements.
10781	MeshVS is not sensitive to width of edges.
11095	MeshVS: Borders of faces are not visible in shrink mode.
11272	Incorrect comments in visualization CDL files.
11543	MeshVS. Vector presentation does not work with default parameters.
11590	[OCC Forum: thread 9366] strange code for texture mapped font
11615	Anisotropic scaling in 3D view.
<b>Application Framework, 16 bug fixes</b>	
ID	Short Description
6340	BinOcaf reader is now compatible with LH3D binary files.
6340	Make BinOcaf reader compatible with LH3D binary files
6493	Binary persistence is included in the TDataStd_ExtStringArray standard attribute.
6604	Detected and improved some exceptions at reading files in the Xml format on Solaris 2.8 with Forte 6.2.
7262	Implementation of Lite OCAF
7724	Compilation error with gcc3.4 at PCDM_RetrievalDriver.hxx
8041	OCAF Browser displays strange symbols
8621	Binary persistence does not work
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
9760	Corrected documentation of TDocStd_MultiTransactionManader
9792	Eliminate dependence of TKDCAF on TKBool, TKBOP, TKFillet,, TKPrim, TKHLR, and TKTopAlgo
10001	New resource unit for XML OCAF
11455	gcc4.0 returns errors at LDOM compilation.
<b>Data Exchange, 22 bug fixes</b>	
ID	Short Description
55	Invalid shape is obtained after reading from a STEP file.
5079	Exception in the translation of IGES files.



5099	Invalid shape is obtained after the translation of attached file.
5953	Wrong STEPControl_Reader constructors.
6283	Unstable work of fixshape on attached shape (different number of warnings).
6284	Exception while reading the file attached to bug 5708. See also bug 5708.
6384	StlAPI_Reader still uses old Sewing algorithm
6491	After sewing in DRAW of the attached shape with tolerance 1000, the resulting shape contains duplicate faces (277 instead of 263).
6497	Bug in IGESControl_Reader::PrintTransferInfo
6508	Reading invalid General Symbol entity from an IGES file causes an exception.
6542	Circle is translated to IGES as NURBS.
6675	Sewing does not work for attached shapes with any tolerance. See bug 6542.
6805	Operation BSplineRestriction produced a shape with null pcurves.
7141	STEP interface crashes exporting large assembly
7735	New toolkits for STEP
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
9490	Increase of tolerance in loop test for IGES
9531	Regression after writing-reading operation
10384	Improvement of the license generator.
<b>Test Harness - Draw, 7 bug fixes</b>	
ID	Short Description
4549	Duplication of updatetolerance command into BRepTest_BasicCommands.cxx and SWDRAW_ShapeTool.cxx
5809	DPLUGIN macro gives a compilation warning on Linux
6132	Wrong ISOS presentation of faces built on a periodic surface in Draw.
7021	Two different draw-commands have the same name
8526	Mode OnlyVisible does not work in the command igesbrep in DRAW.
8662	Implementation of draw command vpick.
9373	Convenient tool for navigating OCAF document under Draw
<b>WOK, 10 bug fixes</b>	
ID	Short Description
5916	The attempts of WOK to compensate time displacement between current workstation and server sometimes make it impossible to work.
6786	WOK does not close unused files, and it leads to errors when a workbench is deleted.
6789	WOK can not generate projects and solutions for MS VC7.1
7286	WOK step xcpp.template does not work on WNT
7302	Creation of possibility to generate *.mak files in order to build whole OpenCascade with MS nmake. Moving of TKjcas to a separate module Wrappers
8362	Compilation errors at obj.cgen on Linux platform
9551	The generation of Makefile.am for TKDXF build incorrect AM_CXXFLAGS variable.
9726	Minor problem in make files
9829	DRAWEXE on WNT is incorrectly linked with option /MT instead of /MD
11579	Incorrect Makefiles for XmlLPlugin.



Shape Healing, 12 bug fixes	
ID	Short Description
5215	Face #1923 can't be split in two faces.
5439	A wire from a face can't be split in ShapeFix_Wire. See also bugs 55 and 5215.
5624	Exception during usage DT_ShapeConvertRev command
5783	Logical error in ShapeFix_TransferParametersProj::TransferRange.
6366	FixShape does not fix an edge with invalid SameRange flag
6555	ShapeFix_Shape modifies valid shape and returns wrong status DONE.
6729	Slow performance of ShapeAnalysis_FreeBounds
6805	Operation BSplineRestriction produces shape with null p-curves.
6810	Duplication of code in ShapeFix_IntersectionTool
7624	Problems in ShapeFix_IntersectionTool
9129	Regress on Mandrake 10
9274	REGRESSION of ShapeAnalysis_FreeBounds on Mandrake
Documentation, 1 bug fix	
ID	Short Description
9463	Update of Draw Test Harness User Guide in part of Data Exchange

**Product Bug Fixes**

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

Open CASCADE Mesh Framework, 18 bug fixes	
ID	Short Description
7615	Adding mesh element with known ID violates MeshIDFactory
7634	Any operation leads to opening of a new window.
7635	Functionality "Show free edges" works wrong
7640	Lost faces after adding them using AddFaceWithID method
7643	Migrate SMDSCaf on Lite-OCAF
7664	Normals are calculated incorrectly during importing of mesh as STL file
7697	Calculation of free boundary of a mesh in SMDSControl should take into account orientation of faces
7905	The OMF sample crash after try to delete Node from mesh (OCCT 5.2.2)
7967	Strange behaviour of OMF sample
8278	Exception after removing a node in a mesh with children
8305	Application crash while deleting interactive object.
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 SMDSCInter_Mesh
9583	Improve OMF by improving NASTRAN reader
9656	New development and repackaging of OMF kernel libraries
9660	Reader of the NASTRAN files can not read files without words "BEGIN BULK"
9800	SMDSTools_STLReader does not read facet normals
Express Mesh, 3 bug fixes	





ID	Short Description
7975	Exception while meshing a shape containing faces without wires
8303	Memory leak in Express Mesh (class QMData_Edge)
8522	Express Mesh exhausts all memory and makes CPU to rest while meshing narrow long face
<b>Surfaces from Scattered Points, 2 bug fixes</b>	
ID	Short Description
8224	Cannot load points from table into PlateFE algorithm.
11226	Tangential constraints don't work correctly
<b>DXF, 1 bug fix</b>	
ID	Short Description
9824	Incorrect translation of DXF file
<b>Canonical Recognition, 1 bug fix</b>	
ID	Short Description
9907	Exception in union faces after canonical recognition
<b>Parasolid, 1 bug fix</b>	
ID	Short Description
10242	Update Parasolid XT reader up to 16 shema

## 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 ShapeAnalysi s\_Wire, ShapeFix\_Wire and others]
- package MgtBRepAbs [the enumeration Tri angl eMode moved to package MgtBRep]
- class BRepTools\_Sewing [class BRepBui lderAPI \_Sewing should be used instead]
- package ShapeInterference [the relevant functionality is covered by Collision Detection product]
- package ShapePlacement
- classes Prs3d\_WFSectionShape, StdPrs\_Tool SectionShape 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 LibCtI : TKernel -> TKXSBase
- Packages Contap, TopBas and TopCnx: TKTopAl go -> TKHLR
- Package TCol Quantify: TKernel -> TKService
- Packages Prs2d, GGraphi c2d, AIS2D, and Sel ect2D: TKV3d -> TKV2d
- Package BRepProj : TKOffset -> TKBool
- Package Moni Tool : TKShHealing -> TKXSBase
- Package TopLoc: TKBRep -> TKMath
- Package BRepAl goAPI : 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 Moni Message (TKShHeal i ng) -> Message (TKernel )
- Package TopOpeBRepApprox (TKBool ) -> BRepApprox (TKTopAI go)
- 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 (Dynam ic, 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, MeshAI go, MeshShape, BRepMesh, and I ntPol y from TKTopAI go toolkit.

As TKSTEP toolkit became too large to handle, including 1548 objects and having a size of more than 10M, it was divided into four toolkits: TKSTEPBase, TKSTEPAttr, TKSTEP209 and TKSTEP.

### **Lite OCAF separation**

#### **Summary.**

- o TDocStd, TDataStd, TDF, TNaming, TFunction packages moved to TKLCAF toolkit;
- o PDF, PDocStd, PNaming, PDataStd, MDF, MDocStd, MDataStd, MNaming packages - moved to TKPLCAF toolkit;
- o StdSchema - several packages removed;
- o PCDMShape package - moved to TKShapeSchema toolkit;
- o PAppStd package - renamed into StdDrivers;
- o PAppStdPlugin executable - renamed into StdPlugin;
- o BinDrivers - several classes removed;
- o BinMDF, BinMDataStd, BinMNaming, BinObjMgt, BinMDocStd, BinTools packages - moved to toolkit TKLBin;
- o XmlDrivers - several classes removed;
- o XmlMDF, XmlMDataStd, XmlMNaming, XmlObjMgt, XmlMDocStd packages - moved to toolkit TKLXml.

#### **Transient OCAF packages and Standard persistence.**

A new package AppStdL has been created, which contains one class: AppStdL\_Appl icati on. This class inherits TDocStd\_Appl icati on class and implements methods Formats() and ResourceName(). This class supports OCC-StdLi te, Bi nLOcaf and Xml LOcaf formats and uses StandardLi te resource file.

Toolkit TKLCAF has been created, which includes packages AppStdL, TDocStd, TDataStd, TDF, TNami ng, TFunc ti on that have been removed from TKCAF toolkit.

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

New StdLSchema schema has been created, which includes packages:

- PDF



- PDataStd
- PNaming
- PDocStd
- PCDM

All packages have been removed from StdSchema except PPrsStd PFunction.

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

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

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

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

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

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

Package PAppStd has been moved to StdDivers, executable PAppStdLPIugin has been renamed to StdLPIugin.

Package PCDMShape has been moved from TKPCAF to TKShapeSchema toolkit. New resource file StandardLite has been created, and the existing plugin file has been adapted to the new format.

### ***Binary Lite persistence.***

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

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

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

All classes have been removed from the BinDivers package.

In package BinDivers two classes have been created: BinDivers\_DocumentRetrievalDriver and BinDivers\_DocumentStorageDriver, which inherit from BinLDivers\_DocumentRetrievalDriver and BinLDivers\_DocumentStorageDriver classes correspondingly. These classes (from package BinDivers) redefine virtual methods AttributeDivers in each class. Calls of BinDivers::AttributeDivers() in AttributeDivers method of each class have been added.

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





Packages Bi nMDF, Bi nMDataStd, Bi nMNami ng, Bi nObj Mgt, Bi nMDocStd, Bi nTool s have been removed from toolkit TKBi n.

New executable Bi nLPI ugi n has been created, similar to Bi nPI ugi n, but uses Bi nLDri vers package.

***Xml Lite persistence.***

Xml persistence has been divided in two parts using same rules as are used for Binary persistence.  
FormatName: **XmlLOcaf**, extension: **xmlI**

