

# Open CASCADE 5.0 Overview

Open CASCADE 5.0 facilitates development of cross-platform applications thanks to a set of new advanced features, add-ons and documentation

*More efficient, more  
productive...  
and still Open Source!*

## What's New in Open CASCADE 5.0 ?

Version 5.0 features new and improved traditional functionality and supplementary add-ons that increase the efficiency of work with Open CASCADE.

### ▪ New Boolean Operations

Boolean Operations algorithms (cut, fuse, common and sectioning) have been substantially rewritten in 5.0 to provide additional reliability and robustness against the previous implementation. This has been especially achieved in the following cases:

- ✓ tangency and overlapping (or same domain) faces;
- ✓ degenerated edges (such as sphere or cone poles);
- ✓ edges shared between the shapes in the operation;
- ✓ more consistent and valid use of internal shape tolerances;
- ✓ reusable cached results of intersection for faster further computations of cut, fuse and common operations.

API has been kept to facilitate easier migration to a new implementation (in particular, classes of the *BRepAlgoAPI* package remain as a main API, now for new algorithms).

Previous implementation has been retained for compatibility reasons and will be gradually withdrawn from Open CASCADE in further releases.

### ▪ New approach to instantiate data collections

With version 5.0 Open CASCADE provides a new approach to create one's own data aggregates (arrays, lists, sequences, etc). Until now developers could only use either WOK, the Open CASCADE development environment, or other libraries (e.g. MFC) to implement custom collections.

Now Open CASCADE 5.0 provides a flexible and powerful mechanism that allows to get rid of this limitation and to instantiate collections using native Open CASCADE features. This can be done as easy as to include just several lines into the code:

*In header file:*

```
DEFINE_BASECOLLECTION (MyPackage_BaseCollPnt, gp_Pnt)
DEFINE_SEQUENCE (MyPackage_SequenceOfPnt,
                MyPackage_BaseCollPnt, gp_Pnt)
```

*In source file:*

```
IMPLEMENT_HSEQUENCE (MyPackage_HSequenceOfPnt)
```

Developers who preferred WOK environment will be able to continue using its techniques of instantiation. Moreover, both techniques can be





mixed in C++ code, and using new data collections in CDL constructions (relevant for WOK only) will be added in future releases.

▪ **LDOM XML parser**

New LDOM XML parser (where LDOM stands for Light Document Object Model) is able to read and write XML files compliant with DOM1 standard as defined by W3C consortium. Note that some features of DOM1 are partially supported.

XML parser is a file engine of the new XML OCAF Persistence that provides storing and retrieving OCAF documents in the XML format (see below for more details about XML OCAF Persistence).

XML parser can also be used separately from XML OCAF Persistence to enable data persistence in the XML format. API classes *LDOM\_XmlReader* and *LDOM\_XmlWriter* provide a complete interface to the parser.



▪ **Numerous improvements and bug corrections**

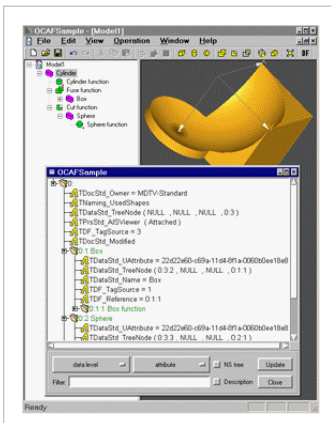
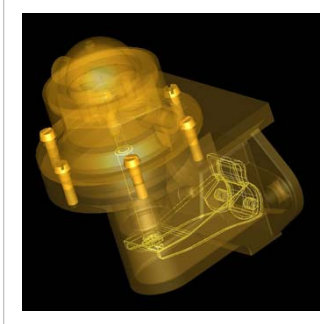
Open CASCADE 5.0 features more than 500 bug fixes over the previous version 4.0 released in November 2001. This version mainly benefits from technical support customers who reported problems in their development. Numerous corrections and improvements have been made within custom development projects conducted by the Open CASCADE company for the clients who ordered these services from us. At last, some problems have been reported by the world-wide Open Source community.

Most corrections and improvements have been made in Modeling Algorithms (about 45%); second position is shared between Data Exchange, Visualization and Modeling Data (15% per each); OCAF corrections constitute about 5% of the total number.

Much attention has also been paid to increasing performance of complex operations especially in visualization, data exchange and shape healing.

For exhaustive enumeration of corrected bugs and added improvements and new features refer to Release Notes supplied with Open CASCADE.

All in all, version 5.0 brings a new powerful platform with industry-proven quality for different types of projects - from open source development to proprietary applications.



▪ **Class browser**

A convenient utility to navigate through the complex class architecture of Open CASCADE, the class browser lets the developers parse class definitions down to the level of class methods and fields.

In addition, for every class the browser produces an inheritance tree, a header file with class definition, parental package, toolkit and module.

The Class browser is supplied as an automatically generated set of HTML files. Any web browser (Netscape, Internet Explorer and others) can be used to navigate over it.

▪ **OCAF browser**

The OCAF document browser is a GUI library used for navigation over the document created with Open CASCADE Application Framework (OCAF). The library can be loaded to explore the document at run-time thereby significantly simplifying the debugging process to ensure correctness of the data model and validity of internal document state.

The browser is supplied with implementation on Fltk, Qt and TclTk graphic toolkits. The source code of the latter can be found in the Draw Test Harness (for instance, in `ros/src/XCAFResources/dftree.tcl`). Fltk and Qt implementations are delivered in the binary form; their source code can be acquired together with a pack of advanced development samples (see below for details).

- **e-Viewer (Standard edition)**

e-Viewer Standard, a web browser's plug-in also available for free download at [www.opencascade.com](http://www.opencascade.com), has been integrated into Open CASCADE 5.0 to ease data sharing over the Internet.

Additionally, you may take advantage of e-Viewer Professional that extends the Standard edition by enabled export capabilities and a wider range of supported formats (including VRML and STL). With Professional edition, you may create on-line converters that will transfer your CAD data directly within your favorite web browser. For more details about e-Viewer Professional please contact [marketing.contact@opencascade.com](mailto:marketing.contact@opencascade.com).

- **New application samples**

A set of numerous samples has been further extended to demonstrate additional capabilities of Open CASCADE. With version 5.0 developers receive real demonstrations on applying graphical textures to 3D models, converting elementary geometry (spheres, cones and so forth) to the NURBS equivalents and tessellating exact geometry with the help of triangulation.

The tutorial application that shows a step-by-step implementation of the first modeling application with Open CASCADE has been now included into the distribution package. Made as a complete cross-platform application with a Qt-based graphical user interface it helps start programming on Open CASCADE faster.

All these samples complement reference documentation and the existing family of samples, which now counts more than 30 small and middle-sized applications. They demonstrate the power of Open CASCADE and ease getting familiar with it for a new user.

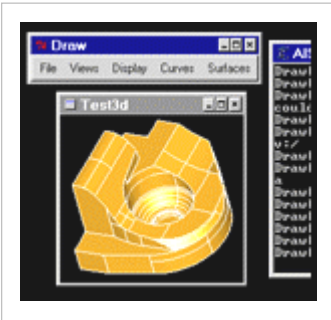
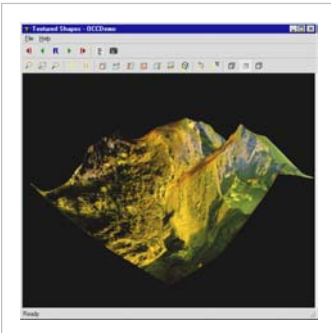
- **Draw Test Harness**

The TCL-based Test Harness provides a flexible, customizable and powerful environment for experimenting with Open CASCADE as well as for testing custom algorithms developed on it.

For several years Test Harness has been mainly oriented to Unix users (due to the organization of our own development process) leaving Windows developers with less functionality. With version 5.0 developers on both platforms may enjoy the same convenience thanks to complete cross-platform consistency. Your own auto-testing scripts will now run seamlessly on all platforms - Sun, Linux and Windows!

The set of available standard commands has also been extended - for instance, about 25 new commands for 2D view manipulation have been added into 5.0.

- **Extended documentation**



Documentation delivered with 5.0 has undergone exhaustive proof-reading and significant extensions have been made to it. This is especially notable in the User's Guides and Reference Manuals on Shape Healing and Data Exchange - modules of constantly growing interest among the users.

Continuing to encourage developers of large projects to use Open CASCADE WOK environment (that provides convenient tools for concurrent development) 5.0 features exhaustive User's Guides on WOK (Workshop Organization Kit) and CDL (Component Definition Language).

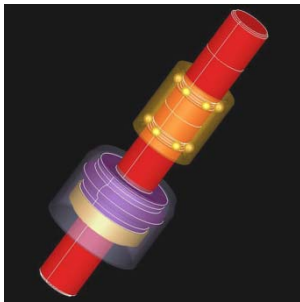
#### Installation procedure



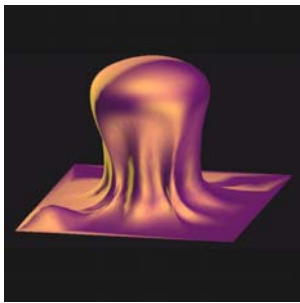
Finally, to help developers enjoy all these exciting features we have created a new convenient installation procedure with a complete cross-platform graphical interface. Sun Solaris and Linux users can now conveniently set-up Open CASCADE components and environment as Windows users.

In this regard, in particular, many efforts have been made to automate installation of WOK environment and to minimize manual efforts required for its customization. Furthermore, installation of e-Viewer is also automatic and the plug-in can be used to navigate through CAD files within your web browser.

The installation process now also gives an overview of major Open CASCADE functionality and new features in the release. However, do not forget to regularly visit our web-site for last-minute updates !



For more details about these new features and improvements please refer to the Release Notes and Reference Documentation supplied with Open CASCADE.



#### Porting to Open CASCADE 5.0

As any major release, Open CASCADE 5.0 requires efforts from developers to rebuild their applications onto a new version. However, all precautions have been taken to keep source compatibility wherever possible without compromising the overall architectural consistency.

When porting applications to 5.0 the following major API changes must be considered:

- ✓ Foundation Classes - the *FSD\_Archive* class that implemented MFC-based serialization on Windows has been removed. Developers who used it may consult the Serialization sample to learn how to port onto 5.0.
- ✓ Modeling Algorithms - package methods of the *BRepAlgoAPI* package have been put into the *BRepAlgo* package.
- ✓ Shape Healing - all packages with *Std* suffixes (such as *ShapeFixStd*, *ShapeAnalysisStd* and so forth) have been removed and their contents have been added to the corresponding package (*ShapeFix*, *ShapeAnalysis* and so on). Respectively, the toolkit *TKShHealingStd* has disappeared.
- ✓ Data Exchange - in a similar way, packages *IGESControlStd* and *STEPControlStd* and their parental toolkits *TKIGESStd* and *TKSTEPStd* have been removed. Complete functionality has been put into units without the *Std* suffix.

## Support of Open CASCADE 4.0

Open CASCADE will provide support of the previous version 4.0 for all users with valid support contracts until expiration of the contract or migration to version 5.0.

### CONTACT US NOW:

#### BY PHONE OR E-MAIL

Call the Open CASCADE sales team at +33 1 69 35 44 52 or send an e-mail to [marketing.contact@opencascade.com](mailto:marketing.contact@opencascade.com).

For more information on Open CASCADE, visit <http://www.opencascade.com>.

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