

# Open CASCADE Technology and Products Version 6.5 Overview

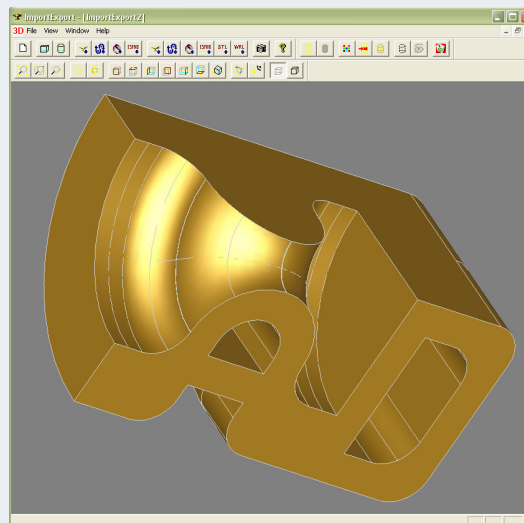
## QUICK FACTS

Open CASCADE Technology is an object-oriented C++ class library designed for development of sophisticated domain-specific design applications on Windows, Linux and Sun platforms.

Open CASCADE Technology is available under the "Open CASCADE Technology Public License". You can download and use it free of charge even for development of your own commercial applications, or ask us to do it for you.

Our numerous clients worldwide have already evaluated the advantages of Open CASCADE Technology.

Open CASCADE Technology 6.5 is a public minor release introducing new features, improvements and bug fixes, over public minor release 6.3.



### Application Framework:

- ◆ Basic OCAF toolkit (TKLCAF) has become independent from OCCT modeling libraries
- ◆ Topological naming in OCAF has been considerably improved

### Foundation classes:

- ◆ A number of improvements has been made in Foundation Classes and Modeling libraries to facilitate the usage of OCCT in multithreaded applications
- ◆ A mechanism to collect statistics and debug memory allocation with OCCT optimized memory manager has been implemented

### Modeling Algorithms:

- ◆ Geometry construction algorithms have been extended by advanced approximation methods
- ◆ Algorithms for line-line, line-plane and plane-plane intersection have been improved
- ◆ New quaternion class for definition and manipulation of 3D rotation operators

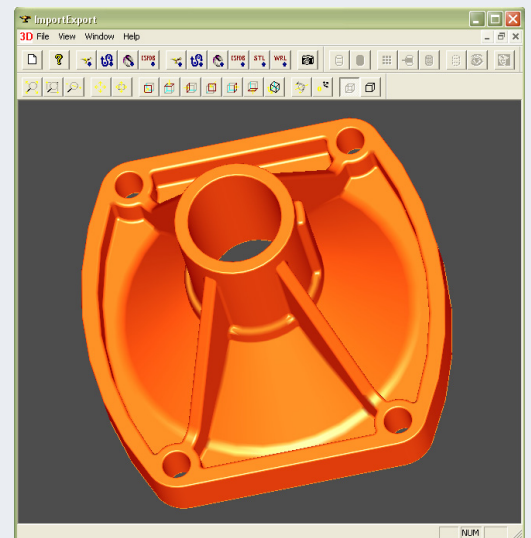
## What's New in Open CASCADE Technology 6.5?

### Visualization:

- ◆ Triangulation algorithms in BRepMesh package have been completely revised for better accuracy, and is now capable to benefit from multi-core CPUs with help of Intel® TBB external library
- ◆ Open CASCADE Technology now uses FreeImage(Plus) third-party library for support of popular image formats
- ◆ New external libraries – freetype and ftgl – are now used for advanced 2D and 3D text

### Data Exchange:

- ◆ STEP translator has been improved to process non-manifold topology in STEP export/import



## OUR SERVICES

### TRAINING

OPEN CASCADE training program allows mastering most of fundamental and advanced functionality, development platform, testing and prototyping techniques. You can get the maximum productivity in using Open CASCADE Technology components, accelerated understanding of the Open CASCADE Technology architecture thus reducing your software development cycle.

### TECHNICAL SUPPORT

Our a-la carte support programs give you a unique opportunity to create flexible combinations of support services as you use them. These programs are based on units that you can use to access any service. Available services vary from individual helpdesk queries to on-site consulting, even training sessions can be ordered within a-la carte programs!

### TURNKEY SOLUTIONS

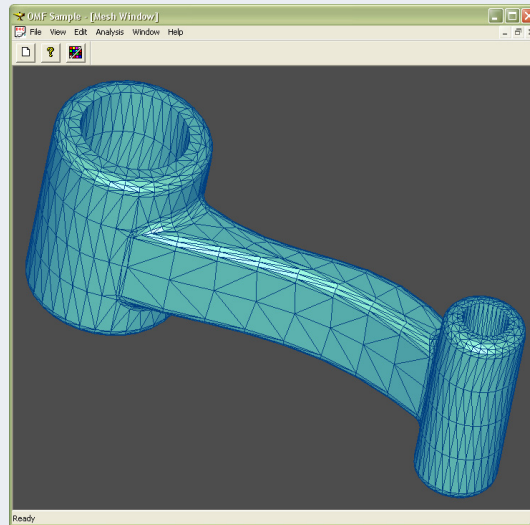
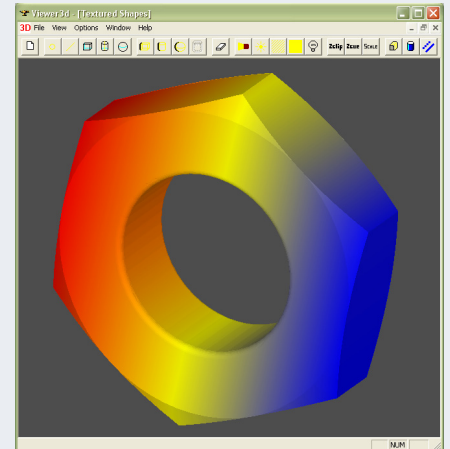
Not enough resources to develop an application on your own? OPEN CASCADE team of professional developers is always ready to develop any specific OCCT-based CAD/CAM/CAE application you may need.

For more information on any of our products or services please visit us on the Web at:

[www.opencascade.com](http://www.opencascade.com) and  
[www.opencascade.org](http://www.opencascade.org)

## What's New in Open CASCADE Products?

- ◆ **New BestFit product**, which computes the transformation minimizing the normal distance between the given cloud of 3d points and the given shape.
- ◆ **Open CASCADE Mesh Framework (OMF) product:** New types of Boolean operations for surface meshes.
- ◆ **Open CASCADE ACIS SAT Interface product:** New methods to read and write SAT data to streams.
- ◆ Products have been extended by DRAW Test Harness plug-ins providing commands useful for testing and initial acquaintance with these libraries



## Constantly Increasing Quality

Open CASCADE Technology 6.5 features about 310 corrections (bug fixes, enhancements and other corrections) over the previous public minor version 6.3 released in September 2008. Most of the problems were found by our customers who reported them in the process of their development projects and had them solved long before this release. A lot of corrections and improvements have been made within our custom development projects conducted for our clients worldwide.

## Supported Platforms

### 32 bit

Windows SEVEN / VISTA SP2 / XP SP3  
Debian 4.0, 5.0  
Mandriva 2008

### 64 bit

Windows SEVEN / VISTA SP2 / XP SP3  
Debian 4.0, 5.0  
Mandriva 2008

For more details about system requirements, please, refer to Technical Requirements at <http://www.opencascade.org/getoccc/require/>

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For more information about OPEN CASCADE SAS, visit <http://www.opencascade.com>