



Open CASCADE Technology 6.2 Overview

Open CASCADE Technology 6.2 is a minor release introducing new features, improvements and bug fixes, over minor release 6.1, and maintenance release 6.1.1.

What's New in Open CASCADE Technology 6.2?

This new minor release of Open CASCADE Technology 6.2 introduces new features and improved traditional functionality along with certain changes over the previous public release 6.1 (released in March 2006) and maintenance release 6.1.1 exclusively available to the customers. With this release users will be able to develop more efficient applications using all the potential of Open CASCADE Technology.

Main improvements in this version:

- Multithread safety has been implemented in OCCT Kernel.
- The exception mechanism on Linux and UNIX has been revised and is now compatible with standard C++ exceptions.
- A new visualization approach based on OpenGL arrays for AIS shapes has been introduced.
- The interactive selection mechanism has been optimized.

Foundation Classes:

- Basic support for multithreading (mutexes, threads) has been provided at the level of TKernel services.
- Implementation of basic OCCT classes such as Standard_Transient and Handle has been optimized.

Visualization:

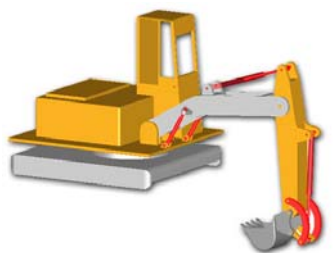
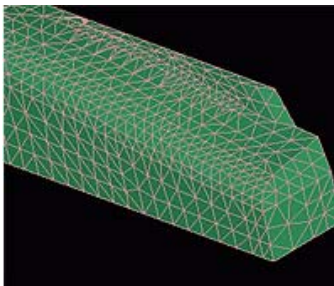
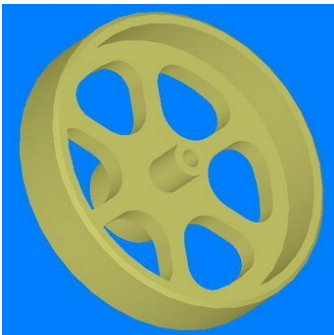
- Interface of MeshVS component extended to allow interpolation of color by single element in accordance with arbitrary color scale (in addition to existing RGB interpolation).
- MeshVS supports now two new types of mesh visualization in shading mode: smooth and flat shading.
- A new drawing attribute in MeshVS allowing to turn on material reflectance on colored 3D representation of data associated with mesh entities, similarly to a shaded mesh representation, to make the presentation look more realistic.

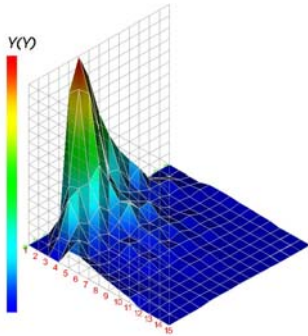
Data Exchange:

- Two new Shape Healing operators added: the first tool is designed for the division of faces in a shape by the maximum allowed area criterion and the second operator is intended for removing inner holes from faces with the area less than the specified minimum area.
- Improved management of warning and fail messages raised during parsing of STEP files.

Documentation:

- Search options have been added to the Reference documentation generated by Doxygen.





Four User Guides have been updated:

- Foundation Classes User Guide,
- Shape Healing User Guide,
- STEP Import / Export User Guide,
- Open CASCADE Mesh Framework User Guide

Support of new platforms, compilers, and build systems:

Additionally to the previous platforms and tools Open CASCADE now supports

- Microsoft Visual Studio 2005;
- SGI 64 bit.

Open CASCADE Products have also been improved with new functionality and bug fixes. The most important are:

Parasolid: Now schema numbers up to 17 are supported; translation of blended edges and colors has been essentially improved.

OMF: Smooth shading visualization mode has been added; numerous other improvements

Surfaces from Scattered Points:

The new Surface Curvature Analysis algorithm facilitates the analysis of the curvature of a surface at its boundaries (edges) and interior.

Constantly Increasing Quality

Numerous improvements and bug corrections

Open CASCADE Technology 6.2 features more than 140 corrections over the previous minor version 6.1 released in March 2006.



Supported platforms

Supported platforms now are:

- 64-bit SUN, DEC, Linux and Windows platforms,
- Windows XP/2000/NT 4.0 SP3,
- Debian Sarge and Woody, Fedora Core 3.0, 4.0,
- Mandrake 7.x, 8.0, 10.1, Mandriva 2006,
- Red Hat 7.1, 8.0,
- Solaris 2.6 + Y2K Patches, Solaris 2.8,
- SGI 64 bit.

For more details about supported platforms please refer to Technical Requirements on <http://www.opencascade.org/getocc/require/>

CONTACT US NOW:

BY PHONE OR E-MAIL

Call the Open CASCADE sales team at +33 1 41 09 42 00 or send an e-mail to marketing.contact@opencascade.com.

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

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