



Open CASCADE Technology 6.1 Overview

Repackaged, faster, with
more functionality!

Open CASCADE Technology 6.1 is a minor release introducing new features, improvements and bug fixes, over minor release 5.2, and maintenance releases 5.2.1, 5.2.2, 5.2.3 and 5.2.4.

What's New in Open CASCADE Technology 6.1?

New Features and Improvements

This new minor release of Open CASCADE Technology 6.1 introduces a lot of new features and improved traditional functionality along with certain changes over the previous public release 5.2 (released in July 2004) and maintenance releases 5.2.x exclusively available to the customers. With this release users will have opportunities to develop more efficient applications using all the potential of Open CASCADE Technology.

Open CASCADE Structure

- The structure of Open CASCADE Technology modules, toolkits and packages has been optimized, i.e. the code has been cleared by removing or separating some obsolete, duplicated, or rarely used items.

Foundation Classes:

- The implementation of memory manager in Open CASCADE has been completely revised. An optimized memory manager has been implemented as a C++ class and now it has become possible to use it in advanced applications requiring separate optimized memory heap.

Visualization:

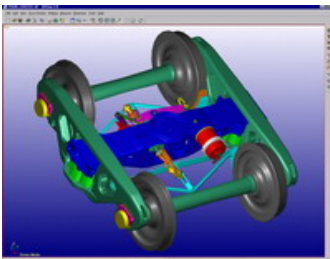
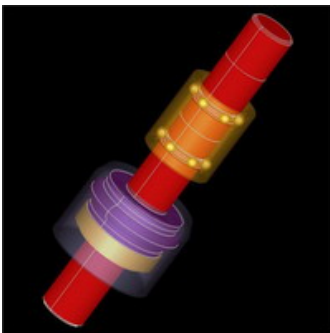
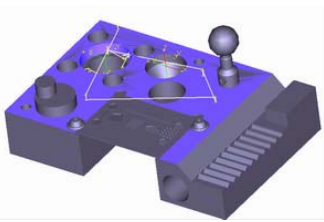
- Colour scale has been implemented in Open CASCADE 3D Viewer;
- The use of texture mapped fonts on Windows is more convenient now;
- Performance has been improved in MeshVS package;
- MeshVS now supports visualization of 3D elements;
- Dump of a 3D viewer scene into vector graphics in PS, EPS and EMF (under MS Windows) formats has been implemented;
- The Foundation Classes User's Guide has been revised

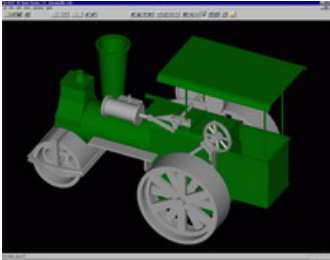
Documentation:

- Starting from version 6.1 Open CASCADE Technology provides documentation generated by *DOXYGEN*, an open source documentation system. Documentation is generated by OCCT modules (OCAF, Foundation Classes, Visualization, etc.).
- Draw test Harness User Guide has been updated in parts of Visualization, OCAF, Topology and Data Exchange.

OCAF:

- *Lite* (simplified) OCAF toolkits have been implemented to save memory and increase the performance in cases when applications do not need visualization.

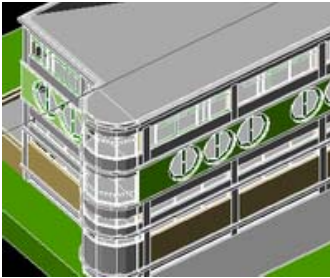




Support of new platforms, compilers, and build systems:

Open CASCADE now supports

- Possibility to build with nmake command on Windows platform;
- Tcl 8.4;
- 64-bit SUN, DEC, Linux and Windows platforms;
- Latest versions of Linux platforms (see 'Supported platforms' below);
- gcc 4.01 and earlier on Linux platform;



Draw Test Harness has been extended with a set of new commands.

Open CASCADE Products have also been improved with new functionality and bug fixes:

- OMF (Open CASCADE Mesh Framework) product has been improved in mesh operations, visualization and data exchange with NASTRAN;
- The Express Mesh algorithm has been improved to avoid hanging and excessive memory consumption when meshing some narrow long faces;
- Parasolid XT reader is updated to support schema numbers up to 16.

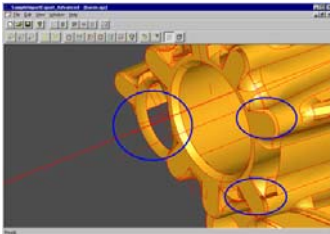


For more details about these new features, improvements and corrected bugs please refer to the Release Notes and Reference Documentation supplied with Open CASCADE Technology. We also recommend to regularly visit our web sites for latest news and updates.

Quality

Numerous improvements and bug corrections

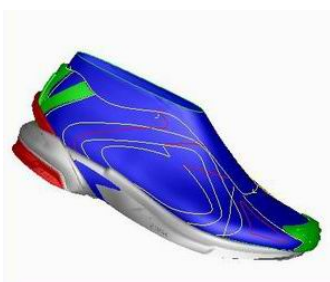
Open CASCADE Technology 6.1 features more than 260 corrections over the previous minor version 5.2 released in July 2004.



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.



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

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

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