

Open CASCADE 5.1 Overview

Open CASCADE 5.1 is a minor release of Open CASCADE introducing new corrections over the previous (major) release and improvements of this powerful platform

*Although minor
but more quality !*

What's New in Open CASCADE 5.1 ?

Version 5.1 is not simply a corrected version of the 5.0 major release of Open CASCADE. It features new and improved traditional functionality as well as certain slight changes. Due to this release Open CASCADE has gained additional power and robustness.

Quality

Numerous improvements and bug corrections

Open CASCADE 5.1 features more than 120 bug fixes over the previous version 5.0 released in May 2003. This version mainly benefits from technical support customers who reported problems in their development and had their problems solved long before this release. 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.

Performance

Visualization improvements are quite evident – especially on complex objects and when working with a significant number of individual objects (e.g. several thousand and more).

- New classes have appeared in the *Graphic3d* package that use OpenGL display lists thus displaying objects much faster;
- Performance has been increased in the animation mode. This became possible due to optional deactivation of text objects when viewing large graphical presentations;
- Performance of multiple selections is now considerably higher than before due to activating/deactivating the highlight mode;
- A new sample – Visualization Performance Meter is now available.

Data Exchange and Shape Healing

- Calculation speed of continuity intervals in the adaptors has increased due to caching of auxiliary methods results.

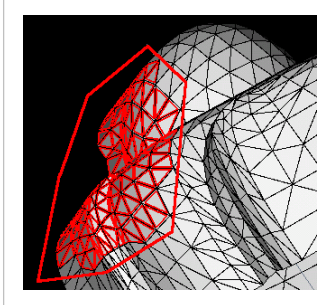
OCAF

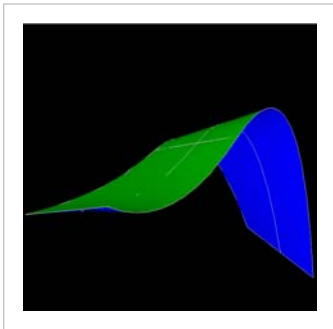
- Improvements have been achieved in the Memory allocation mechanism for large OCAF documents and Undo/Redo operations;
- Performance has been increased in exploring documents.

Improvements

Modeling Algorithms appears to be a module introducing most new features and improvements in this release:

- *New Boolean Operations*: now even more operational
A new feature of Open CASCADE 5.0 – New Boolean Operations has been enhanced by a history mechanism. This mechanism significantly eases the shape-editing process especially in feature-based modeling applications;





- Algorithms for modeling pipe surfaces have been extended to support some new cases;
- Computation of parameter(s) of a given point on a 2D or 3D curve or a surface is now more convenient due to an auxiliary new class in the *GeomLib* package.

Visualization

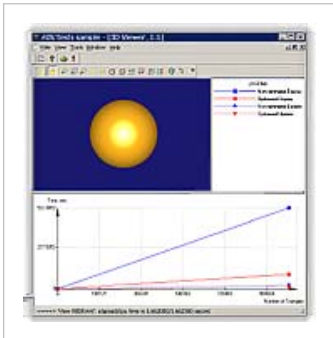
- Different properties can now be set to different sides of a surface;
- 3d Viewer is now more fun to work with: it now supports background images;
- Polygonal selection of interactive objects has been enabled;

Data exchange and Shape Healing

- Gaps can be fixed and small edges can be merged in “floating” wires (those lying by themselves and not bounding any faces);
- Faces based on Revolution Surfaces (type 120) and Tabulated Cylinder (type122) are handled keeping the parametric space;
- In addition to previously supported protocols (AP 203 and AP 214) initial support of a new one has been added: Application Protocol 209 for finite-element analysis (parsing and formatting files without mapping to objects).

OCAF

- Support of Open/Save operations return status has been added. Now it is possible to trace persistence failures inside the program to provide the user with proper information about disk, file and file content problems;



Rebuilding tools

Rebuilding limitations have been overcome in Open CASCADE 5.1. Numerous improvements have been introduced to the **build scripts** used to rebuilt Open CASCADE. Rebuilding of Open CASCADE modules has been improved by means of MS Visual Studio project files on Windows platform, and GNU tools (*autoconf*, *automake* and *libtool*) on Unix.

Installation Procedure

The **installation process** with its new visual line 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, improvements and corrected bugs please refer to the Release Notes and Reference Documentation supplied with Open CASCADE.



Supported platforms

Supported platform now are Windows 2000/NT, Linux Mandrake 7.X, 8.0 and Red Hat 7.X, Sun Solaris 2.6. Open CASCADE no longer supports Windows 98 and Red Hat 6.0 operating systems. If you are using one of the above we encourage you to port onto one of the systems we continue to support.

For more details about supported platforms please refer to Technical Requirements on www.opencascade.com

CONTACT US NOW:

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Call the Open CASCADE sales team at +33 (0)1 69 35 44 52 or send an e-mail to marketing.contact@opencascade.com.
For more information on Open CASCADE, visit <http://www.opencascade.com>.

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