Open CASCADE Technology 6.3 Overview

Open CASCADE Technology 6.3 is a minor release introducing new features, improvements and bug fixes, over minor release 6.2, and maintenance release 6.2.1.

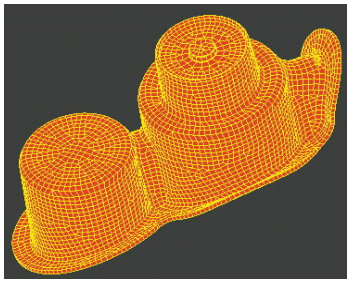
What's New in Open CASCADE Technology 6.3?

This new minor release of Open CASCADE Technology 6.3 introduces new features and improved traditional functionality along with certain changes over the previous public release 6.2 (released in March 2007) and maintenance release 6.2.1 exclusively available to the customers. Open CASCADE Technology has become more powerful and robust.

Main improvements in this version:

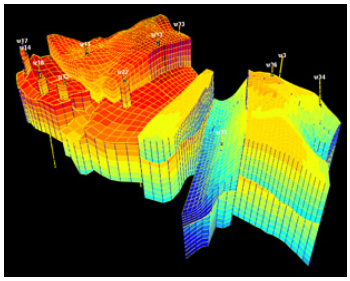
Modeling Algorithms:

- Open CASCADE tessellation algorithms (used for building triangulation of shapes in visualization) have been unified and improved.
- Progress indication for reading and saving of brep files has been implemented



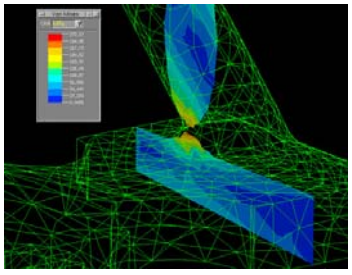
Visualization:

- Selection mechanism in MeshVS has been improved to support a great number of selectable mesh entities (500K and more) with a good response time.
- Simplified visualization of vectors has been implemented in MeshVS package.
- Support of mesh group selection has been introduced in MeshVS package.
- The possibility to record an AVI video stream from OpenGl code has been implemented
- A new visualization library NIS (New Interactive Service) has been implemented



Application Framework:

- A new mode of storing transaction delta has been implemented in some OCAF attributes.
- Function Mechanism of Open CASCADE Application Framework (OCAF) has been extended.
- New version of the OCAF binary persistence format has been implemented.
- Objects serving as an intermediate layer between OCAF data attributes and high-level (e.g. GUI) code has been implemented



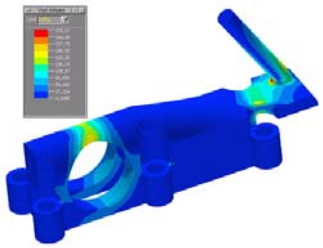
Data Exchange:

- The functionality of reading/writing VRML2.0 files has been implemented in OCCT.

Foundation Classes

- Next step in thread-safety: protection against concurrent construction / destruction of Handle objects

User Guides have been updated:



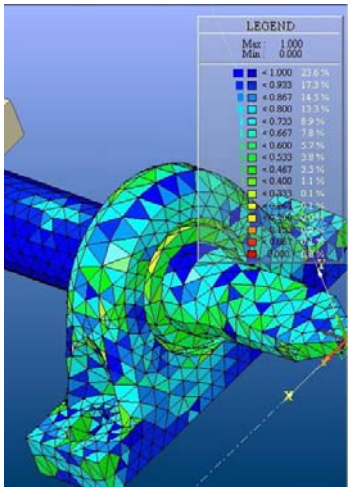
- Visualization User Guide,

New User Guides have been written and added:

- Voxels User Guide
- OCAF TObj package User Guide
- OCAF Function Mechanism User Guide

Support of new platforms, compilers, and build systems:

Additionally to the previous platforms Open CASCADE Technology now supports:
Windows Vista, Mandriva 2006, Mandriva 2007, Mandriva 2008, Debian Etch 4.0



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

ACIS: *Reading of ACIS SAT versions 10.0 - 14.0*

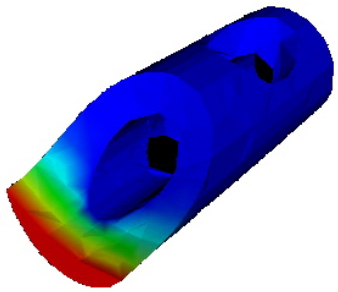
OMF: *Improved tools for reading and writing NASTRAN files in OMF*

Constantly Increasing Quality

Numerous improvements and bug corrections

Open CASCADE Technology 6.3 features over **170** corrections over the previous minor version 6.2 released in March 2007. Most of the problems were found by our technical support customers who reported them in their development 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 VISTA/XP SP2/2000/NT 4.0 SP6,
- Debian Sarge , Woody,
- Fedora Core 3.0, 4.0, 6.0
- Mandrake 7.x, 8.0, 10.1,
- Mandriva 2006,2007,2008
- Scientific Linux4.2,4.3
- Red Hat 7.1, 8.0,
- Red Hat Enterprise 4.0,
- Solaris 2.6 + Y2K Patches
- SGI

64 bit

- Windows XP SP2,
- Debian Etch,
- Mandriva 2006,2008
- Red Hat Enterprise 4.0
- Solaris 2.8,
- SGI

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

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