

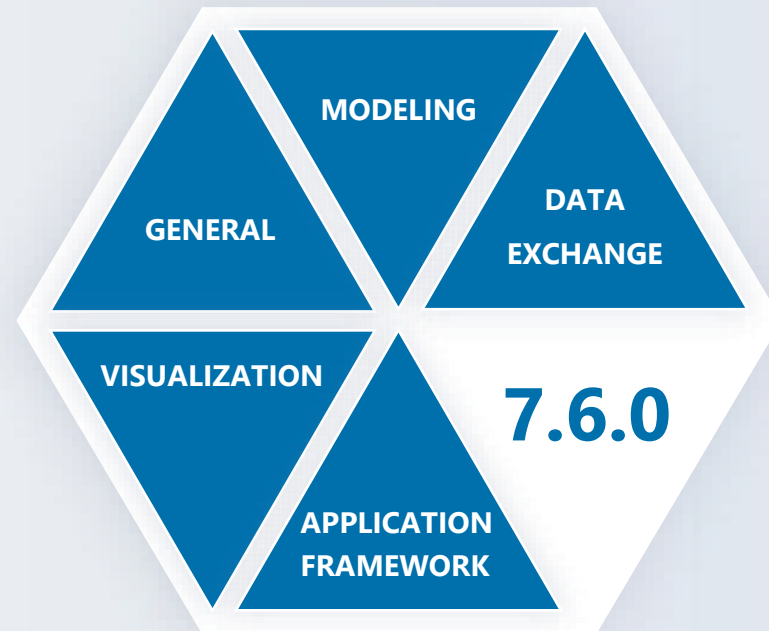


Status update



Open CASCADE Technology 7.6.0 Release

Open CASCADE Technology 7.6.0 contains more than 400 improvements and corrections over the previous release. Fixes touch all aspects of OCCT including general fixes, modeling, data exchange, visualization, and application framework.





7.6.0 Release: general

- Improved configurability – now Xlib, Tk and FreeType are optional
- Dropped support of Visual Studio 2008
- Forbid scaled locations in TopoDS_Shape
- BREP format (text and binary) update



7.6.0 Release: modeling

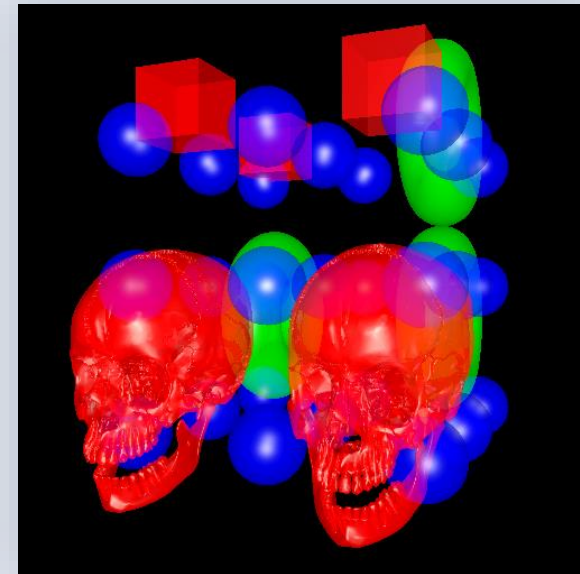
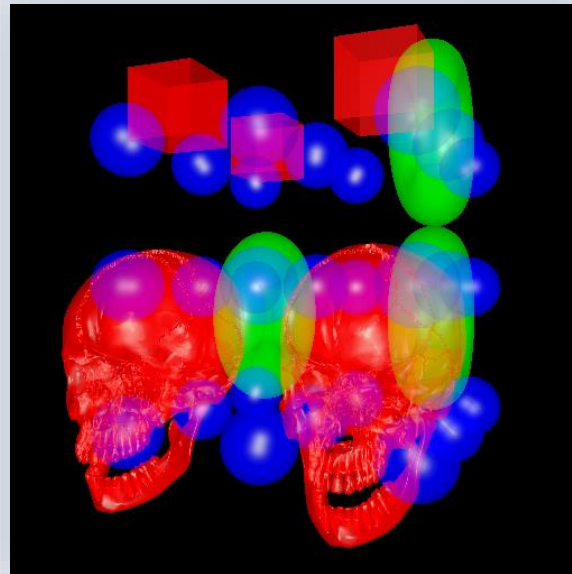
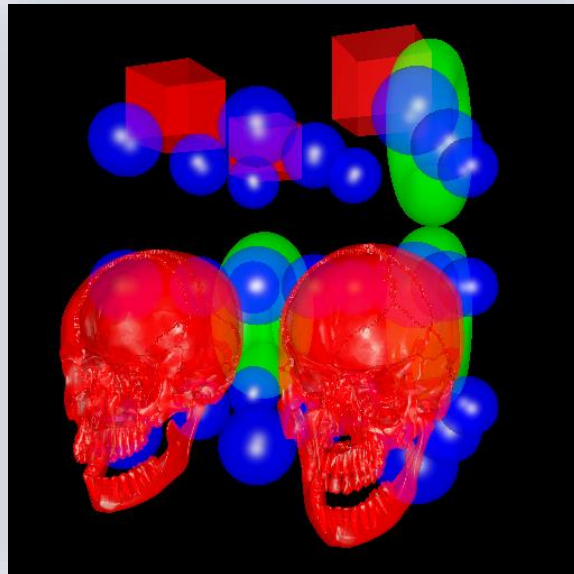
- Extrema - parallel execution, memory-related optimizations, support of trimmed inputs (curve / curve)
- Progress indicator – added to Boolean operations, offset algorithm, and Extrema
- Get rid of old Booleans API
- Improvements in Poly package

Extrema (specific use case)	OCCT 7.5.0	OCCT 7.6.0
Peak memory, MiB	2900	152
Duration, 1 thread, s	155	109
Duration, 8 threads, s	N/A	26



7.6.0 Release: visualization

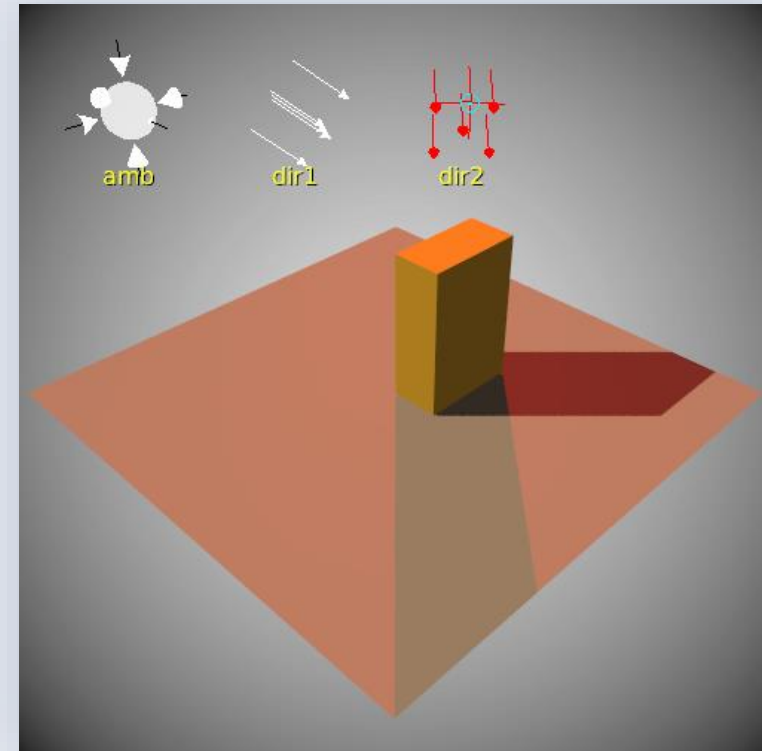
New method for accurate order independent transparency (depth peeling), compatible with wide range of platforms (including WebGL 2.0).





7.6.0 Release: visualization

- Simple shadow-casting from directional light source
- Elliptical style background
- New sensitive classes for spherical, conical, and cylindrical objects without triangulation
- Improved compatibility with OpenGL ES / WebGL implementations





7.6.0 Release: data exchange

- glTF – reading of Draco compressed files
- glTF – new export options for optimizing structure
- STEP – entities to read kinematics
- OBJ – writer is added

Sample glTF model (10M tris, 383 parts)	Uncompressed	Draco- compressed
File size, MiB	186	19
Load time (slow Internet), s	80	10



7.6.0 Release: application framework

- Extract sub-assembly
- Partial loading of OCAF documents, see also [blog post](#)
- Better versioning of OCAF documents
- Writing documents in old formats




Plans

- Drop support of pre-C++11 compilers – [link](#) to forum tread
- Keep working on robustness and stability in modeling
- AIS interactive object interface improvements and extensions
- Thread reentrancy of STEP translator
- Support of tessellated geometry in STEP translator
- Improve and extend user guides and tutorials



Decreasing entry threshold

- Free trainings publication
- Novice guide
- Samples restructure
- Highlighting code snippets in documentation

- ▼ Open CASCADE Technology 
- ▶ Introduction
- ▼ Tutorials and Samples
 - ▼ Tutorials and Demos
 - ▶ **Novice Guide**
 - ▶ Draw: Demo Scripts
 - ▶ Modeling: Bottle Tutorial
 - ▶ OCAF: Usage Tutorial
 - ▶ OCAF: Function Mechanism
 - ▶ Sample Projects



Training: Topology

FREE

Open CASCADE TECHNOLOGY



Training: Geometry

FREE

Open CASCADE TECHNOLOGY



Training: Preliminaries

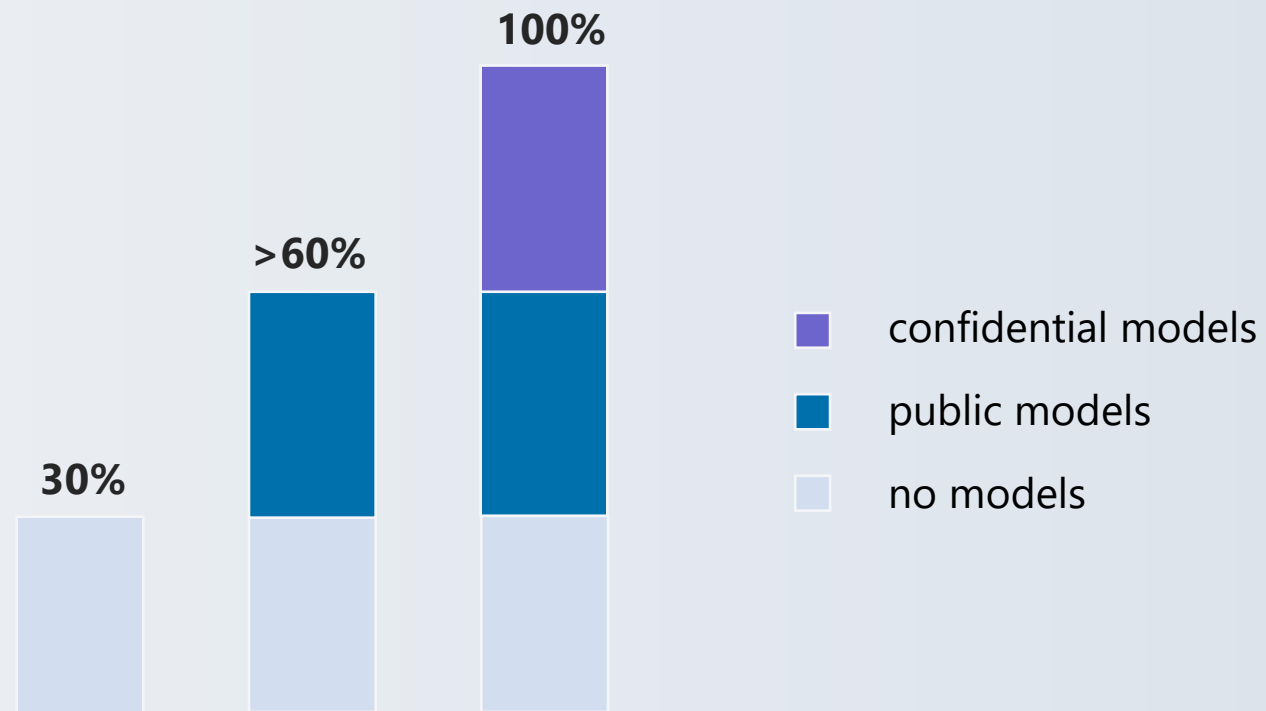
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Open CASCADE TECHNOLOGY



OCCT Testing Dataset

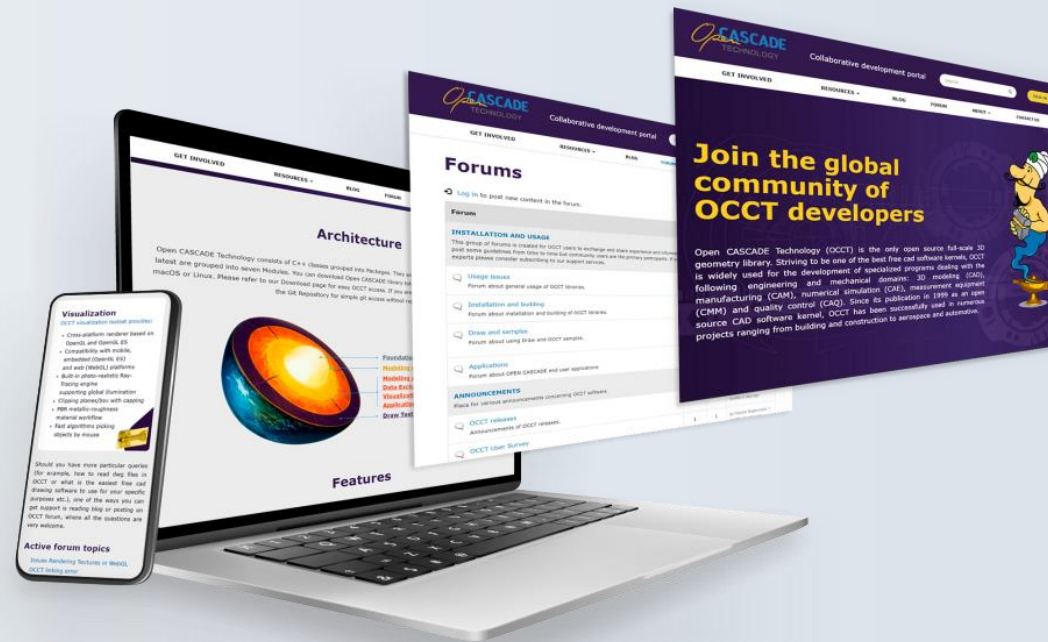
Testing dataset of more than 2500 shapes is published and updated, increasing OCCT test coverage up to around 60%.





New website for Open CASCADE Technology

- Modern look and mobile-friendly interface
- Single Sign-On (SSO)
- New forum structure
- More contribution opportunities
- New community sections





OCCT Projects & Products Marketplace

OCCT Projects & Products Marketplace is a single access point to many projects, that helps to ease the entry into technology and provide a wider look at OCCT capabilities:

- Variety of OCCT-based projects for >40 industrial sectors
- 31 projects are added, more to come
- Share experience with the community by requesting to list your project!





OCCT Research & Science Publications

OCCT is commonly used in [academic and research spheres](#) by high-profile organizations all around the world.

Open Cascade Research & Science Publications Listing includes more than [600 articles](#) and research projects conducted by [400+ organizations](#) in [48 countries](#).

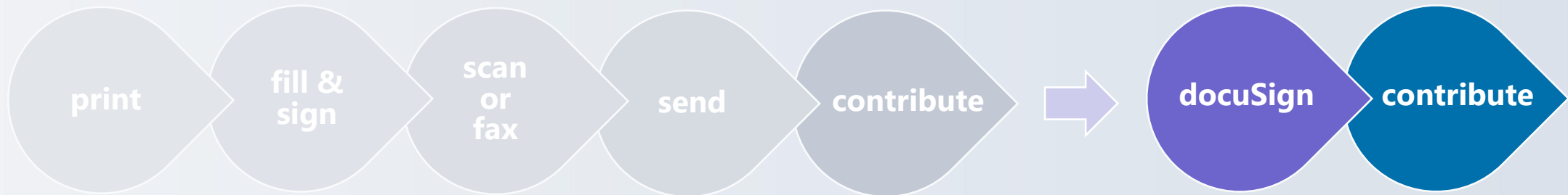
[Explore](#) the collection of research publications or [share](#) your article with the community.





Plans for 2022

- Fully digital CLA signing process option in addition to the current workflow



- Technical Blog featuring articles from external authors
- OCCT rebranding process
- Further developer website UX improvements



Useful links

- 1 Feel free to download OCCT and testing dataset:
<https://dev.opencascade.org/release/>
- 2 Documentation:
<https://dev.opencascade.org/doc/overview/html/>
- 3 Open CASCADE Technology 7.6.0 press release:
<https://www.opencascade.com/open-cascade-technology-7-6-0-released/>
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