Open CASCADE Technology

Version 7.7.0

Release Notes

November 3, 2022

Open CASCADE Technology

Overview

Open CASCADE Technology 7.7.0 provides about 250 improvements and corrections over the previous release 7.6.0.

Highlights

General

- Improved compatibility with C++17/C++20 compilers
- Dropped support of pre-C++11 compilers

Modeling

- New functionality is implemented, which could verify the input shape to be placed on a canonical geometry with the given tolerance. If the input shape is a

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face or a shell, it could be verified to be close enough to Plane, Cylinder, Cone or Sphere. If the input shape is an edge or a wire, it could be verified to be close to Line, Circle or Ellipse as well as lying on one of the analytical surfaces above.
- Introduced new tool BRepLib_PointCloudShape generating a point set for a topological shape.
- New option in BRepOffsetAPI_MakeOffset - approximation of input contours by ones consisting of 2D circular arcs and 2D linear segments only, it provides more stable work of 2D offset algorithm.

Visualization

- Introduced new interface for creating V3d_View as subviews of another V3d_View.
- Added smoothing to row interlaced stereoscopic output.
- Added word-wrapping option to Font_TextFormatter.
- Added support of a wide color window buffer format (10bit per component / 30bit RGB).
- Added MSAA anti-aliasing support when using WebGL 2.0.
- Introduced skydome generation feature V3d_View::BackgroundSkydome().

Mesh

- BRepMesh works too long and produces many free nodes on a valid face problems are resolved.
- Meshing the shape no longer takes too long and visualization problems are corrected.
- Wrong shading display of thrusections is fixed.
- Rendering issue when using deviation coefficient of low value is resolved.
- Mesher no longer produce 'bad' result for extruded spline with given deviation coefficient.
- Holes in triangulation with large linear deflection are removed.
- Broken triangulation on pipe shape is fixed.

Data Exchange

- STEP translator now supports tessellated presentations.
- Transformation tools BRepBuilderAPI_Transform/BRepBuilderAPI_Copy now handle properly tessellated presentations.
- glTF Writer - added support of Draco compression.
- Introduced DEWrapper - a unified interface to Data Exchange connectors.
- Introduced tool XCAFDoc_Editor::RescaleGeometry() for scaling geometry in XCAF document.

Configuration

- SONAME is now configurable in CMake and includes minor version in addition to major by default.
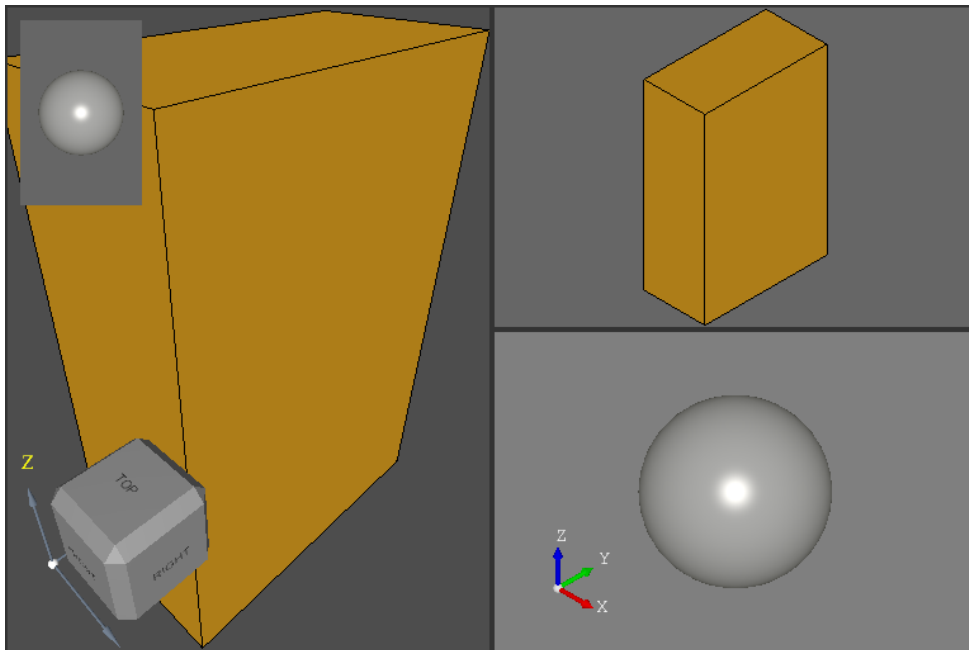
Documentation

- Improved samples / tutorials documentation.
- Introduced new "AIS: Custom Presentation" tutorial.

New Features

Visualization - improved multi-view support

V3d_View implementation has been extended with a new interface allowing to define views as subviews of another V3d_View serving as a composer. This interface provides multi-view support to platforms that do not support creation of multiple native windows or disallow/restrict sharing of OpenGL resources between them (like Android or WebAssembly). It also simplifies implementing features like thumbnail/split views on any platforms.

Upgrade

Added a new chapter "Upgrade to OCCT 7.7.0" (see OCCT html documentation: "Build, Debug and Upgrade / Upgrade from older OCCT versions / Upgrade to OCCT 7.7.0").