Open CASCADE Technology and Products ver. 6.5.5 Maintenance Release

Release Notes

Overview

Open CASCADE Technology and Products version 6.5.5 is a maintenance release, which includes about **30** improvements and bug fixes over maintenance release 6.5.4.

Version 6.5.5 is source and binary compatible with version 6.5.4 by top-level API and ABI; no porting is required. However, the applications that use internal class BOP_SolidSolid and classes from IntPolyh package must be recompiled to run with this version.

Highlights

- **BREP Format Description White Paper has been added**

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Modifications

Modeling Data

23475	*Summary:* Wrong result of `Geom_BSpline_Surface::IsVClosed()` The algorithm finding if the b-spline surface is closed has been corrected to compare two isolines on the opposite bounds of the surface rather than opposite poles of the surface definition. The algorithm comparing two b-spline curves for equality is implemented as new method `IsEqual ()` of the class `Geom_BSplineCurve`.

Modeling Algorithms

23530	*Summary:* Invalid face is not recognized in Draw command "`checkshape`" Functions `BrepCheck_Wire::IsDistanceIn3Dtolerance` and `BrepCheck_Wire::IsDistanceIn2Dtolerance` have been implemented to determinate closure in 2D and 3D.
23535	*Summary:* Exception in `BSplCLib::BuildCache()` Method `IntCurve_IntCurveCurveGen::InternalCompositePerform` has been refactored to reduce overhead for recursive calls.
23548	*Summary:* Boolean operation between two faces fails Low-level data structures in IntPolyh package has been refactored to avoid limitations leading to failure of intersection algorithm in some cases.
23552	*Summary:* Projection algorithm produces wrong results with default tolerance value `Precision::PApproximation()` tolerance has been replaced by `Precision::Confusion()` in `ProjLib_ProjectedCurve` class to improve the work of Projection algorithm. Method `ChFi3d_Builder::Compute()` now inspects the new faces to provide `sameparameter` if necessary.
23576	*Summary:* Intersection algorithm produces trimmed circle with illegal parametric range Method `IntTools_LineConstructor::TreatCircle` has been added to provide correct processing of curves of type Circle and Ellipse. It replaces the obsolete method `IntTools_LineConstructor::PeriodicLine`. Function `IntPatch_ImpImpIntersection::IntCyCy` has been modified to correctly process the case with two ellipses to provide two ellipses instead of four.

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23587	*Summary:* Draw operation "2dintersect" cannot find intersection point of two intersecting 2d curves. New method `IntCurve_IntPolyPolyGen::FindIntersect` has been implemented to find intersection between two curves. It returns false if some points of polygon were replaced on line and exact point of intersection was not found for case when point of intersection was found.
23603	*Summary:* Boolean operation between two edges fails Inner class `IntTools_ComparatorCurve` has been added in `IntTools_EdgeEdge` to check whether the curves are the same or not. Additionally, the procedure of curve retrieval has been changed to take into account trimmed and offset curves in method `IntTools_ComparatorCurve::IsSameBSplineCurve()`.
23604	*Summary:* Uninitialized variables in debug mode Directives `#ifndef DEB ... #else ... #endif` have been removed wherever the variables are initialized only in release mode to avoid errors in debug mode.
23606	*Summary:* Invalid result of pipe operation New method `ShareFaces()`, which shares coincident faces in the resulting shape has been added in `BRepFill_Pipe` class.
23643	*Summary:* Intersection algorithm produces B-Spline curve in case of coaxial cones Processing of coaxial cones has been added in method `IntPatch_Intersection::Perform`.
23644	*Summary:* Intersection between coaxial cones produces exception Processing of coaxial cones with coinciding apices has been added in method `IntAna_QuadQuadGeo::Perform`.
23676	*Summary:* Intersection between edge and a face produces extra vertex A typo has been corrected in method `IntTools_EdgeFace::CheckData()`.
23699	*Summary:* Exception raised during intersection between two surfaces Methods `IntPolyh_MaillageAffinage::TriangleComparePSP()` and `IntPolyh_MaillageAffinage::TriangleCompare()` have been improved to disallow intersection between triangles with degenerated links.
23703	*Summary:* Projection algorithm produces incomplete 2D-Curve The algorithm of comparison between two curves in method `ProjLib_CompProjectedCurve::InitialPoint` has been corrected.
23704	*Summary:* The program hang-up during intersection of two edges Processing of B-Spline or Bezier curve intersections with other curves has been improved in method `IntTools_EdgeEdge::IsSameCurves()`.

23711	*Summary:* Incorrect result done by fuse operation The following improvements have been introduced so that fuse operation could process cases, when arguments have common closed boundaries. • Fields myRefObject and myRefTool have been added to store working solids of arguments. • Method BOP_SolidSolid::PrepareReferences has been added to build working solids of arguments. • Method BOP_SolidSolid::TakeOnSplit has been modified to process same-domain faces.
23716	*Summary:* Boolean Fuse between two faces fails 2D boundaries of sphere-based faces have been extended to provide lasting results of 2D classifications for the parts of intersection curves in method IntTools_FaceFace::CorrectSurfaceBoundaries.
23732	*Summary:* Exception raised during intersection between two surfaces Processing of degenerated zones on surfaces has been improved in method IntPolyh_MaillageAffinage::FillArrayOfPnt.
23733	*Summary:* PCurve for edge on face creation failure Flag myIsMin is now computed for each solution for ellipse extrema in class Extrema_ExtPElC. Parameter adjustment has been implemented in class Extrema_ExtPRevS for periodic curves (circles and ellipses) to keep the parameter inside the parametric range of the curve.
23738	*Summary:* Boolean Fuse between two faces fails New method IntAna_QuadQuadGeo::RefineDir allows refining aDir, containing 1, (or -1) as component and to provide unit length of the direction.

Visualization

23705	*Summary:* Isoline in the AIS viewer is not trimmed Tolerance for Hatch_Hatcher in method StdPrs_WFDeflectionRestrictedFace::Add now depends on minimal distance between face boundary points.

Data Exchange

23567	*Summary:* Wrong STEP unit is returned by STEPControl_Reader.FileUnits() function The problem with returned wrong values has been solved by passing unit object to STEPControl_Reader::getSiName() function directly.

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Mesh

23473	*Summary:* Top face of imported STEP part is missing Constraints for complex internal wires have been corrected in method `BRepMesh_Delaun::RemovePivotTriangles`.

Products

ACIS Import

22755	Summary: Exception translating a SAT file The procedure of names reading from a SAT file has been corrected.

DXF Import

21829	Summary: Small improvement for `TranslatePolyline` `DxfData_TranslatePolyline.cxx` has been modified to provide correct translation.
23553	*Summary:* Exception is raised on reading of the attached corrupted DXF file An `If` statement to check object before calling its methods has been added in `DxfSat_TranslateBody.cxx` to avoid application crash.

DXF Export

23449	*Summary:* DXF file Witten by the OCC DXF Export can not be reread
	
	The algorithms of reading and writing periodic B-Splines have been corrected:
	  ▪ on writing, they are converted to a non-periodic form;
	  ▪ on reading, periodicity is restored after construction of b-spline.
	
	Special processing of b-splines with short list of knots, which are supported by DXF True View but is not read by other tools (Rhinoceros, DS DraftSight, ProE) has been added. It allows loading curves, even if the shape is not always well reproduced.

23507	*Summary:* Double call of `DXFControl_Writer::TransferShape` leads to writing an incorrect file Double call has been eliminated in method `DXFControl_Writer::TransferShape`.

Advanced Samples

23565	*Summary:* Ship Builder update Environment and documentation for Ship Builder module have been updated.
23621	*Summary:* Incorrect parsing of the importing file name if path name contains dot symbol. Files with names containing "." symbol now can be correctly imported. A warning has been added for the case when it is not possible to import a file.
23719	*Summary:* Light update Readme.txt with instructions about building MFC samples has been added.

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Supported Platforms and Pre-requisites

Open CASCADE Technology is supported on Windows Intel and Linux Intel platforms.

The table below lists the product versions used by OCCT and its system requirements.

Linux Operating System	32/64-bit: Debian 4.0, Mandriva 2010*
Windows Operating System	32/64-bit: MS Windows SEVEN / VISTA SP2 /XP SP3
Minimum memory	512 Mb, 1 Gb recommended
Free disk space (complete installation)	650 Mb of disk space, or 1,4 Gb if installed with reference documentation
Minimum swap space	500 Mb
Video card	**GeForce** The following versions of GeForce drivers are recommended: *For Linux:* 64-bit Version: 100.14.19 or later 32-bit Version: 100.14.19 or later *For Windows:* Version 266.58 WHQL or later is recommended: http://www.nvidia.com/Download/index.aspx
Graphic library	OpenGL 1.1+
C++	*For Linux:* GNU gcc 4.0. - 4.3.2. *For Windows:* Microsoft Visual Studio .NET 2005 SP1 with all security updates Microsoft Visual Studio .NET 2008 SP1** Microsoft Visual Studio .NET 2010 Microsoft Visual Studio .NET 2012
TCL (for testing tools)	*For Linux:* Tcltk 8.5 http://www.tcl.tk/software/tcltk/8.5.html *For Windows:* ActiveTcl 8.5 http://www.activestate.com/activetcl/downloads
Qt (for demonstration tools)	Qt 4.6.2 http://qt.nokia.com/downloads
Freetype (OCCT Text rendering)	freetype-2.4.10 http://sourceforge.net/projects/freetype/files/
Ftgl (OCCT Text rendering)	ftgl-2.1.3 http://sourceforge.net/projects/ftgl/files/
FreeImage*** (Support of common graphic formats)	FreeImage 3.14.1 http://sourceforge.net/projects/freeimage/files/Source%20Distribution/
gl2ps*** (Export of OCCT viewer contents to vector graphic file)	gl2ps-1.3.5 http://geuz.org/gl2ps/
TBB*** (Tool for parallelized version of BRepMesh component)	tbb30_018oss http://www.threadingbuildingblocks.org/

- * Mandriva 2010 is a permanently tested platform.
- ** The official release of OCCT for Windows contains libraries built with VC++ 2010.
- *** This product is optional.