



Open CASCADE Technology and Products ver. 6.5.2 Maintenance Release

Release Notes

Overview

Open CASCADE Technology and Products version 6.5.2 is a maintenance release, which includes **92** new features, improvements and bug fixes over maintenance release 6.5.1.

Version **6.5.2** is binary incompatible with the previous versions of Open CASCADE Technology and Products, so applications linked against a previous version must be recompiled to run with this Version 6.5.2.

Highlights

- ➔ **Redesign of Print operation – new Tile algorithm**
- ➔ **Support of 2-byte character strings in visualization**
- ➔ **Improvement of BRepMesh triangulator and Intersection algorithms**
- ➔ **Update of ACIS Import, DXF Import and DXF Export products**





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Modifications

Foundation Classes

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| 22293 | <p><i>Summary:</i> Incorrect memory allocation for <code>NCollection_IncAllocator</code> on Windows 64 bit</p> <p>Memory allocation problems have been fixed by casting to <code>size_t</code> instead of <code>unsigned long</code> in <code>NCollection_IncAllocator.cxx</code>.</p> |
| 22544 | <p><i>Summary:</i> Fixed <code>IsEqual</code> from Standard</p> <p>The global functions <code>IsEqual</code> for <code>Standard_CString</code> and <code>Standard_ExtString</code> have been considered confusing and removed. The user is able to define own functions if needed.</p> |
| 22562 | <p><i>Summary:</i> Helper C++/CLI class to encapsulate C++ classes</p> <p>Working in .NET environment with OCCT or other C++ libraries, it can be often useful to store an instance of C++ object as a field of C++/CLI class. This is not allowed directly, but can be achieved through indirection (by pointer).</p> <p>The new template class <code>NCollection_Haft</code> allows encapsulating a C++ class instance in a C++/CLI class that can be used in other C++/CLI classes and provides transparent access to the encapsulated C++ instance.</p> <p>The new C++/CLI class is defined in a header file with extension <code>.h</code> to distinguish it from normal C++ classes (with extension <code>.hxx</code>).</p> |

Modeling Data

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| 22038 | <p><i>Summary:</i> Suppress "!!! No Error" message in <code>BRepTools::Write</code> in debug mode</p> <p>An obsolete warning message has been removed from <code>BRepTools</code> class.</p> |
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Modeling Algorithms

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| 21578 | <p><i>Summary:</i> The attached shape is not visualized in the shaded mode</p> <p><code>BRepMesh</code> triangulator has been improved to build triangulations on some cases of faces with self-intersecting wires.</p> <ul style="list-style-type: none"> ▪ Newly introduced self-intersection check in <code>BRepMesh_Classifier</code> calculates the area of the main loop and the area between the first and the second intersection points. Self-intersection is detected when both areas are large or if the wire has only one intersection point. ▪ An obsolete constructor that takes <code>BRepMesh_IndexedMapOfVertex</code> as a parameter was deleted and three local functions were isolated from the class to simplify the code. |
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| <p>21757 21858 22502 22507 22652</p> | <p><i>Summary:</i> Visualization hangs on the attached face</p> <p>The issue with use of inconsistent tolerances for checking point coincidence in parametric space of faces in BRepMesh triangulator has been fixed.</p> <p>To address this problem, a new data structure BRepMesh_VertexTool has been introduced to insert points in UV space with tolerance Precision::Confusion() and to enable scaling of U and V axes, which is necessary to avoid hangs on high-resolution shapes.</p> |
| <p>22109</p> | <p><i>Summary:</i> Boolean operation Common fails.</p> <p>Processing of the cases with faces containing internal edge(s) has been improved in method BOP_Refiner::DoInternalS()</p> <p>Internal edges added to wire-edges are set twice with opposite orientation to provide right path-tracing in method BOP_ShellSolid::AddSplitPartsINOUT (const Standard_Integer nF1, BOP_WireEdgeSet& aWES).</p> |
| <p>22296</p> | <p><i>Summary:</i> BRepSweep_MakeRevol produces incorrect toroidal based face</p> <p>The creation algorithm for pure toroidal surface from the surface of revolution in class Adaptor3d_SurfaceOfRevolUtilon has been brought in correspondence with the definition of toroidal surface in Geom_ToroidalSurface.cdl.</p> |
| <p>22302</p> | <p><i>Summary:</i> BRepMesh_Incremental Mesh calls for each face of shape on vdisplay</p> <p>The calculation of shape triangulation in wireframe mode has been corrected.</p> <p>Boolean useTriangulation parameter has been added in method BrepBndLib::Add to calculate triangulation is if a shape has no triangulation or if its deflection is higher than the new value. This parameter allows creating a bounding box for a shape with various types of deflection: if useTriangulation is true and the triangulation exists it is taken into account for calculating the bounding box, otherwise a bounding box is calculated with standard deflection.</p> |
| <p>22306</p> | <p><i>Summary:</i> Fuse between cylinder and sphere produces not valid result</p> <p>The methods DecompositionOfWire and FindPoint have been corrected in class IntTools_FaceFace to improve Fuse results.</p> |
| <p>22310</p> | <p><i>Summary:</i> Boolean operation Common fails</p> <p>Processing of intersection between two faces in method BOPTools_PaveFiller::PutPaveOnCurve has been changed taking into account the existence of "stick" vertices, obtained from the interferences of lower class.</p> |
| <p>22313</p> | <p><i>Summary:</i> Bug in shading mode with the attached shape</p> <p>BRepMesh_FastDiscret and BRepMesh_Incremental Mesh have been revised to eliminate artifacts appearing in shading mode on some shapes.</p> |
| <p>22322</p> | <p><i>Summary:</i> Improvement of Extrema performance</p> <p>Standard_EXPORT directive required by C# Wrapper has been added to the private method BRepExtrema_DistanceSS::Perform.</p> |





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| 22356 | <p><i>Summary:</i> The result of Fuse is not valid for two straight edges</p> <p>Processing of intersection between the two edges has been changed in method <code>IntTools_EdgeEdge::ComputeLineLine()</code> to take into account the existence of shared vertices of the edges.</p> |
| 22383 22699 | <p><i>Summary:</i> Four and more points on curve</p> <p><code>GCPnts_TangentialDeflection</code> class has been modified to reduce the number of points on curve. If the curvature is insignificant, the curve counts as a line and at least two points are used.</p> |
| 22409 | <p><i>Summary:</i> Boolean operation Cut fails</p> <p>The tolerance value for a vertex has been improved in method <code>BOPTools_PCurveMaker::Do()</code> to take into account the existence of 3D-curves and p-curves of adjacent edges.</p> |
| 22435 | <p><i>Summary:</i> Wrong tolerance value for intersection curves</p> <p>Processing of intersection between the faces based on <code>Cylinder</code> and <code>SurfaceOfRevolution</code> surfaces has been improved in <code>IntTools_FaceFace::ComputeTolReached3d()</code> method. The tolerance of the obtained intersection 3D curves has been modified in accordance with the real values that are computed as the maximum distance between the corresponding 3D points from 3D-curves and 2D-curves.</p> |
| 22489 | <p><i>Summary:</i> <code>BRepClass3d_SolidClassifier::PerformInfinitePoint()</code> gives wrong result on the given solid</p> <p><code>BRepClass3d_SolidClassifier::PerformInfinitePoint()</code> has been modified to improve processing of solids.</p> |
| 22494 | <p><i>Summary:</i> <code>BRepClass_FaceClassifier</code> bug</p> <p>The criteria of intersection between 2d-Line and 2D-Circle have been refined in class <code>IntCurve_IntConicConic</code>. The case of two solutions has been defined more exactly by the criteria of closeness of the solutions in 2D space</p> |
| 22519 | <p><i>Summary:</i> Boolean operation gives different results</p> <p><code>IntPatch_PrmPrmIntersection</code> class has been fixed to avoid the dependence of the result of intersection between two faces upon the order of faces.</p> |
| 22564 | <p><i>Summary:</i> <code>BrepMesh_Classifier</code> improvements</p> <p>Copy/paste from the constructor, check on natural restriction from this class, and use of the field from <code>BrepMesh_FastDiscreteFace</code> have been removed from <code>BrepMesh_Classifier</code> class.</p> |
| 22579 | <p><i>Summary:</i> Improving thread-safety of <code>GeomFill</code></p> <p>Static variables have been removed from <code>GeomFill.cxx</code> to improve thread safety.</p> |





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| 22584 | <p><i>Summary:</i> Optimization of BrepMesh_DiscretFactory to cache registered factory methods</p> <p>Previously plugin searching routines (including library loading and exported method retrieving) were performed on each request of meshing algorithms using factory interface (<code>::Discret()</code> method). This made often access to any non-internal meshing algorithm inefficient.</p> <p>Now plugin availability is determined during <code>::SetDefault()</code> method call (which also returns the success state) and cached for fast access.</p> |
| 22586 | <p><i>Summary:</i> ShapeFix_FixSmallFace mistakenly removes a non-small pipe-like face with a seam edge</p> <p>The class ShapeAnalysis_CheckSmallFace has been modified to avoid incorrect face removal.</p> |
| 22588 | <p><i>Summary:</i> Revolution with negative volume in some cases.</p> <p>The computation algorithm for the value of resulting solid orientation has been refined in BRepSweep_Rotation to avoid the uncertainty due to the value of a vector product.</p> |
| 22589 | <p><i>Summary:</i> Uninitialized memory access issues</p> <p>Uninitialized memory access problems have been fixed by initialization of all fields in classes <code>IntRes2d_Transition</code>, <code>Domain</code>; <code>Intf_SectionPoint</code>; <code>Adapto3d_CurveOnSurface</code> and <code>Bnd_Box2d</code> to avoid crashes of <code>GeomPlate</code> algorithm.</p> |
| 22593 | <p><i>Summary:</i> Data races in Poly</p> <p>Incorrect static variables have been removed from <code>Poly_Connect</code> class.</p> |
| 22595 | <p><i>Summary:</i> <code>gp_Mat</code> constructors incompletely initialize memory</p> <p>Constructors of <code>gp_XYZ.Ixx</code> and <code>gp_Mat.Ixx</code> have fixed to avoid using indirect memory reference <code>((Standard_Real*)&matrix[0][0])[i]</code> to access elements of the member field <code>matrix[3][3]</code>, which resulted in uninitialized memory on some compilers due to erroneous optimizations.</p> |
| 22600 | <p><i>Summary:</i> TBB has to be disabled in BrepMesh due to data races</p> <p>Previously mesh parallelization was controlled in an unobvious manner via <code>(HAVE_TBB macros + Standard::IsReentrant())</code>. Now this control is given directly to BrepMesh classes as a non-global field accessed via <code>SetParallel()</code> and <code>IsParallel()</code> public methods. Thus the user can explicitly choose the necessary behavior.</p> <p>BrepMesh_DiscretFactory::Discret() method now returns <code>Handle</code> instead of pointer to private field in the global Factory. Plugin connection is now performed in method <code>SetDefault()</code> and cached, thus <code>Discret()</code> method has become offloaded and just creates a new meshing tool instance.</p> <p>New method <code>SetParallelDefault()</code> has been added to <code>BrepMesh_IncrementalMesh</code> to control default parallelization flag for Factory. <code>DRAWEXE</code> command <code>mpparallel</code> allows switching this flag on fly.</p> |





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| 22610 | <p><i>Summary:</i> The algorithm <code>GeomAPI_ProjectPointOnSurf</code> produces wrong results</p> <p><code>Extrema_GenExtPS.cxx</code> has been modified to improve the solution using a smaller sub-grid near the start point</p> |
| 22646 | <p><i>Summary:</i> Error in algorithms <code>BRepOffsetAPI_ThruSections</code> and <code>BRepOffsetAPI_MakePipeShell</code></p> <p>Obsolete degenerated edges have been removed from the internal data structure of <code>BRepFill_CompatibleWires</code> class.</p> |
| 22653 | <p><i>Summary:</i> Bad performance of Open CASCADE libraries used by Partition Algorithm</p> <p>The following new methods have been introduced to improve the performance of OCCT libraries used by the Partition algorithm:</p> <ul style="list-style-type: none"> ▪ <code>TCollection_DataMap::Find1(const TheKey& K)</code> – returns the address of <code>Item</code> of the key <code><K></code> or NULL if <code>K</code> is not in the map. The object is not modifiable. ▪ <code>TCollection_DataMap::ChangeFind1(const TheKey& K)</code> – returns the address of <code>Item</code> of the key <code><K></code> or NULL if <code>K</code> is not in the map. The object is modifiable. <p>The methods allow to perform the two operations: checking (<code>IsBound(...)</code>) and finding (<code>Find(...)</code> and <code>ChangeFind(...)</code>) at once.</p> <ul style="list-style-type: none"> ▪ <code>TCollection_IndexedDataMap::FindFromKey1(const TheKey& K1)</code> – returns the address of <code>Item</code> of the key <code><K></code> or NULL if <code>K</code> is not in the map. The object is not modifiable. ▪ <code>TCollection_IndexedDataMap::ChangeFromKey1(const TheKey& K1)</code> – returns the address of <code>Item</code> of the key <code><K></code> or NULL if <code>K</code> is not in the map. The object is modifiable. <p>The methods allow checking (<code>Contains(...)</code>) and finding (<code>FindFromKey(...)</code> and <code>ChangeFromKey(...)</code>) at once.</p> <p>These new methods of <code>TCollection_IndexedDataMap</code> class are used by <code>IntTools_Context</code> class.</p> <p>The following changes have been implemented:</p> <ul style="list-style-type: none"> ▪ The computation of the distances in 3D has been replaced by the computation of the square distances in class <code>IntTools_Tool</code> wherever possible. ▪ The execution order in method <code>Bnd_Box::IsOut(const Bnd_Box& Other)</code> has been changed to provide processing of the trivial boxes first. |
| 22678 | <p><i>Summary:</i> Bad result of the Cut operation.</p> <p>The procedure of putting paves on the intersection curves in method <code>BOPTools_PaveFiller::PutPavesOnCurves()</code> has been changed to take into account the vanishing intersection curves between faces based on toroidal surfaces.</p> |
| 22688 | <p><i>Summary:</i> Syntax error in file <code>BrepOffset_Inter2d.cxx</code>: wrong usage of brackets in comparison</p> <p>Additional brackets have been inserted in <code>BrepOffset_Inter2d.cxx</code> in conformity with the precedence rule.</p> |
| 22689 | <p><i>Summary:</i> Infinite loop in static function <code>TRIM_INFINIT_FACE</code></p> <p>A misprint has been fixed in <code>BRepExtrema_DistanceSS.cxx</code>.</p> |





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| 22692 | <p><i>Summary:</i> Wrong function called in <code>Extrema_CurveTool::IsRational</code></p> <p>The method <code>Extrema_CurveTool::IsRational()</code> has been fixed to correctly call <code>Degree()</code> function.</p> |
| 22695 | <p><i>Summary:</i> <code>BRepFillletAPI_MakeFilllet2d</code> algorithm crashes when face edges become punctual</p> <p>The comparison operators have been corrected in some cases in <code>ChFi2d_Builder</code> (<code>></code> replaced by <code>>=</code> and <code><</code> by <code><=</code>).</p> |
| 22720 | <p><i>Summary:</i> Wrong intersection point between sphere and plane</p> <p>The check of intersection points has been added in method <code>IntPatch_ImpIntersection::Perform()</code> to analyze whether the intersection point is within the 2D-domain of the surface or not. Only the intersection points within the 2D-domains of both surfaces are added to the resulting sequence</p> <p>The static function <code>intersection(Draw_Interpreter& di, Standard_Integer n, const char** a)</code> provides graphical display of the intersection points for the case of intersection of two surfaces.</p> |
| 22723 | <p><i>Summary:</i> Wrong intersection curve between cylinder and plane</p> <p>The method <code>IntTools_FaceFace::PrepareLines3D(const Standard_Boolean bToSplit)</code> has been extended with the flag <code><bToSplit></code> to provide optional possibility to obtain non-split closed 3D-curves:</p> <ul style="list-style-type: none"> ▪ If <code><bToSplit></code> is true, the closed 3D-curves will be split. ▪ If <code><bToSplit></code> is false, the closed 3D-curves remain untouched. <p>The class <code>IntPatch_ImpIntersection</code> has been modified to provide correct processing of the cases of intersection between a plane and cylinder (sphere, cone) when the intersection curve is a circle. The orientation of the local coordinate system of the circle is adjusted with the local coordinate system of the cylinder (sphere, cone).</p> |
| 22725 | <p><i>Summary:</i> <code>BRepOffsetAPI_ThruSections</code> algorithm produces twisted surface on some wires</p> <p>Method <code>BRepFill_CompatibleWires::ComputeOrigin(const Standard_Boolean polar)</code> has been modified to search more correctly for a non-twisting order of edges in sections.</p> |
| 22733 | <p><i>Summary:</i> Empty result for the extrema between a circle and a line.</p> <p>The values of directions in method <code>Extrema_ExtElC::Extrema_ExtElC</code> have been refined to provide unit length.</p> |





Visualization

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| <p>7691</p> | <p><i>Summary:</i> Wrong hidden lines computed by HLRBRep_PolyAlgo and HLRBRep_PolyHLRToShape</p> <p>Hidden line removal algorithm has been improved for parallel view type in class HLRBRep_PolyAlgo.</p> <p>To display the result of HLR algorithm for shape object in v3d view a new DRAW command <code>vcomputeHLR</code> has been added in <code>ViewerTest</code> package. The syntax of this command is as follows: <code>vcomputeHLR ShapeName HLRName [EyeX EyeY EyeZ LookX LookY LookZ]</code>, where <code>ShapeName</code> is the name of initial shape being processed by HLR algorithm; <code>HLRName</code> is the name of interactive object that will be created to display result of HLR algorithm in v3d view; <code>{EyeX, EyeY, EyeZ}</code>, <code>{LookX, LookY, LookZ}</code> is the projector position and projection direction. If the position and direction are not defined, the command takes the position and the view direction of an active v3d view.</p> |
| <p>21981</p> | <p><i>Summary:</i> Additional callback before redraw procedure</p> <ul style="list-style-type: none"> ▪ New callback executed before redrawing the underlayer has been added to <code>call_togl_redraw</code>, <code>call_togl_redraw_area</code> and <code>call_togl_print</code> functions in <code>OpenGL</code> package. ▪ New bit flag <code>OCC_PRE_REDRAW</code> has been added in <code>Aspect</code> package to inform about callback execution before the underlayer is redrawn. The reason value of <code>Aspect_GraphicCallbackStruct</code> is bitwise summed with <code>OCC_PRE_REDRAW</code> to inform that the callback is executed before redrawing the underlayer. ▪ New bit mask <code>OCC_REDRAW_ADDITIONAL_CALLBACKS</code> can be used in callback methods to check if any bit flags, for example <code>OCC_PRE_REDRAW</code> and <code>OCC_PRE_OVERLAY</code>, have been added to "reason" value of <code>Aspect_GraphicCallbackStruct</code>. ▪ <code>NIS_View</code> callback method has been extended with check for "reason" value. The callback will be performed only at the end of redrawing. |
| <p>22149</p> | <p><i>Summary:</i> Strings with Japanese characters cannot be displayed in 3D viewer</p> <p>Draw command '<code>vdrawtext</code>' has been modified to allow displaying in OCCT 3D viewer multi-byte character strings, for example, for Japanese symbols, which previously were always converted to ASCII.</p> <p>The command now accepts up to 15 parameters: <code>vdrawtext name, X, Y, Z, R, G, B, h_align, v_align, ang, zoom, height, Aspect, FONT, isMultiByte</code>.</p> <p><code>FONT</code> can be given with spaces using brackets, e. g. <code>{Times New Roman}</code></p> <p><code>isMultiByte</code> is optional, 0 by default, if the value is 1, it activates a special interpretation of 'name' string. To encode extended characters in the 'name' string use two-byte codes, for example: <code>\x30\x42</code> - encodes value 0x3042.</p> |
| <p>22330</p> | <p><i>Summary:</i> Compilation errors from '<code>clang</code>' on OS X</p> <p><code>Visual3d_TransientManager.cxx</code> has been corrected to avoid compilation errors on OS X.</p> |





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| <p>22337</p> | <p><i>Summary:</i> V3d_View : Print crashes</p> <p>The printing operation under Windows has been completely redesigned. The following improvements and changes have been implemented:</p> <ul style="list-style-type: none"> ▪ The function <code>call_togl_print</code> has been modified to use the off-screen buffer (frame buffer object) for rendering that makes the result independent from the current 3D view dimensions. ▪ It has become possible to choose between two print algorithms that define how the 3D scene is mapped to the print area when the maximum dimensions of the frame buffer are smaller than the dimensions of the print area. Tile algorithm uses the newly implemented <code>TileSplit</code> algorithm, which allows covering the whole print area by rendering multiple parts of the viewer. Theoretically, this means that any printing resolution could be achieved without stretching, to avoid image quality loss. This algorithm prints parts of the 3D scene in the maximum possible resolution into an off-screen buffer, and then copies them to the print area, covering it by tiles. Stretch algorithm draws the 3D scene using the maximum supported frame buffer size and then stretches the image to the whole printing area. The stretching functionality provided by <code>FreeImage</code> is applied with the bicubic interpolation. ▪ New parameter <code>const int printmode</code> has been added to <code>call_togl_print</code>. This method now returns <code>True/False</code> to indicate if the print data has been successfully sent to the printer. ▪ <code>V3d_View::Print</code> method returns <code>Standard_True/Standard_False</code> to indicate if the print data has been successfully sent to the printer. ▪ The new class <code>OpenGL_PrinterContext</code> provides the specific printing information to the OpenGL redraw procedure. The instance of this class can be created only in <code>call_togl_print</code> method and can be obtained by the <code>OpenGL_PrinterContext::GetPrinterContext()</code> method. The printer context is associated with the OpenGL rendering context for better robustness. ▪ <code>TelSetViewIndex</code>, <code>call_togl_redraw_layer2d</code> and <code>OpenGL_TextRender::RenderText</code> methods have been modified to use printing information provided by <code>OpenGL_PrinterContext</code> instance. |
| <p>22455</p> | <p><i>Summary:</i> Make it possible to call methods of V3d_View like <code>Convert()</code> inside the context of <code>Redraw()</code></p> <p>The method <code>V3d_View::SetWindow()</code> has been modified to assign the field <code>MyWindow</code> before calling <code>Redraw()</code>, otherwise <code>MyWindow</code> is left uninitialized during redrawing, which can lead to an exception at the attempt to call some method, for example, <code>Convert()</code> from within callback.</p> |
| <p>22577</p> | <p><i>Summary:</i> Thread-safety/reentrability improvements</p> <p>Static variables have been removed from <code>AIS_GlobalStatus.cxx</code> and <code>AIS_InteractiveContext.cxx</code>. Additionally, <code>Standard_Atomic_*</code> functions have been improved to return operation result.</p> |
| <p>22583</p> | <p><i>Summary:</i> <code>Graphi_c3d_ArrayOfPolygons</code> is not displayed if it has both bounds and edges</p> <p><code>Graphi_c3d_ArrayOfPrimitives::IsValid()</code> checking procedure for primitive arrays has been corrected to pass arrays that are correct and have defined vertices, "bounds" and "edges" together.</p> |





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| 22583 | <p>Redundant logical blocks in <code>Graphic3d_ArrayOfPrimitives::IsValid</code> have been removed.</p> <p>DRAW command "vdrawparray" has been added to test all types of primitive arrays. This command draws primitives array of defined type by its vertices, bounds, edges passed in form of a string as input arguments.</p> |
| 22650 | <p><i>Summary:</i> Exception in Primitive Array during Redisplay of Presentable Object</p> <p>Previously the handles to primitive arrays in <code>Graphic3d</code> classes were removed without notifying <code>TKOpenGL</code>.</p> <p>Now <code>Graphic3d_Structure</code> controls and removes on destruction its references in <code>TKOpenGL</code>, so the handles are deleted at the right moment. The following procedure has been implemented:</p> <ul style="list-style-type: none"> ▪ <code>Graphic3d_Structure</code> sends commands to its <code>Graphic3d_Groups</code> on Remove and Clear operations; ▪ <code>Graphic3d_Group</code> clears its references in Destroy method if it is still visualized and might have shared primitive arrays in <code>TKOpenGL</code> display; ▪ <code>Graphic3d_Group</code> sends command to <code>TKOpenGL</code> to remove primitive array pointers in the methods <code>Graphic3d_Group::RemovePrimitiveArray</code> and <code>Graphic3d_Group::RemovePrimitiveArrays</code>; <p>The following features have been implemented in <code>OpenGL</code> package:</p> <ul style="list-style-type: none"> ▪ New interface method <code>OpenGL_GraphicDriver::RemovePrimitiveArray</code> is called by <code>Graphic3d_Group</code> when the group deletes the <code>PrimitiveArray</code> from its list; ▪ New method <code>call_togl_parray_remove</code> has been added to find primitives array in the <code>TKOpenGL</code> structures by its group and pointer and remove references to it. |
| 22716 | <p><i>Summary:</i> Method <code>Aspect_GradientBackground::SetBgGradientFillMethod</code> is not implemented</p> <p>Obsolete declaration has been removed from <code>Aspect_GradientBackground</code> class.</p> |
| 22738 | <p><i>Summary:</i> Exception on destruction of <code>Graphic3d_Structure</code></p> <p><code>Graphic3d_Structure::Remove</code> operation has been corrected to avoid exception with <code>MMGT_OPT=0</code>.</p> |

Application Framework

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| 22573 | <p><i>Summary:</i> Extend the range of status values returned by the method <code>TDocStd_Application::SaveAs</code></p> <p>The enumeration <code>CDF_StoreStatus</code> has been renamed into <code>PCDM_StoreStatus</code> and extended with additional items to inform about the cause of document saving failure (e.g., when the destination disk is full).</p> |
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Data Exchange

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| 7707 | <p><i>Summary:</i> Unused static function CheckSubShape in XCAFDoc_ShapeTool . cxx</p> <p>The obsolete static function CheckSubShape has been removed.</p> |
| 22436 | <p><i>Summary:</i> Extra compound is created when importing a non-manifold topology from STEP file</p> <p>STEPControl_ActorRead has been slightly adjusted to explode the NMSSR compound if a non-manifold mode is set for the reader (read. step. nonmanifold == 1).</p> |
| 22492 | <p><i>Summary:</i> Scaled sphere (Solid with BSplineSurface) is wrongly exported in STEP.</p> <p>A condition on b-spline surfaces has been added to StepToTopoDS_TranslateFace : Init to make processing of VERTEX_LOOP STEP entity applicable not only for spherical surfaces but also for b-spline surfaces.</p> |
| 22568 | <p><i>Summary:</i> Declaring method as protected in XCAFApp_Application</p> <p>The methods of XCAFApp_Application and XCAFPrs_AISObject have become protected instead of private, which enables subclassing these classes.</p> |
| 22572 | <p><i>Summary:</i> Support of progress indicator in RWStl and optimization of reading ASCII.</p> <p>Progress indicator has been added to Write...() and Read...() methods. Reallocation of string buffers when writing STL files in ASCII mode has been reduced</p> |
| 22578 | <p><i>Summary:</i> Stl Mesh_Mesh migration from private to protected data fields</p> <p>The data fields of Stl Mesh_Mesh have become protected (subclasses cannot access/change them).</p> |

Shape Healing

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| 20485 | <p><i>Summary:</i> Shape Processing - SplittedUsedFace produces wrong shape</p> <p>The problem caused by wrong parameterization used for points on the curve has been fixed in ShapeUpgrade_SplittedCurve3d.</p> |
| 22576 | <p><i>Summary:</i> ShapeFix_Edge : SameParameter() now retains the original 3D interval</p> <p>Previously a periodic curve with 3D range lying outside of a period was adjusted to fit into the range in method ShapeFix_Edge : SameParameter(). Thus copyedge used different ranges than the original edge and if its pcurves were copied back, ranges and pcurves became inconsistent and new redundant edges were inserted.</p> |
| 22587 | <p><i>Summary:</i> Problem with a missing seam fixer</p> <p>Method ShapeFix_ComposeShell : SplittedByLine has been adjusted to take care of notched edges while splitting wire segments.</p> |





Draw

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| 22686 | <p><i>Summary:</i> Some Draw commands do not return necessary information in Draw_Interpreter, but write in cout.</p> <p>Some commands in MeshTest_CheckTopology and MeshTest_PluginCommands have been modified to correctly write information in dout instead of cout.</p> |
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Documentation

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| 22520 | <p><i>Summary:</i> Update Visualization User's Guide</p> <p>The Visualization User's Guide has been revised to reflect the current state of OCCT visualization/selection API.</p> |
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Building tools

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| | <p><i>Summary:</i> Building OCCT and Products with Makefiles building procedure</p> <p>Since OCCT 6.5.2 before configure step you have to launch the script build_configure to generate configure and Makefiles files.</p> |
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Products

Parasolid

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| <p>21611 21613</p> | <p><i>Summary:</i> Some X_T files are not read</p> <p>XT-reader has been adjusted to skip non-numeric real values and process them as zero values. The changes have been integrated in XtData package.</p> |
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ACIS Import

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| <p>7266</p> | <p><i>Summary:</i> Solid with unit location read from ACIS file.</p> <p>The class <code>Aci sData_Aci sToCascade</code> has been modified to allow the SAT-reader to correctly process affine transformations, which do not represent rotations and translations. A special processing routine has been introduced for such transformations.</p> |
| <p>22538</p> | <p><i>Summary:</i> Invalid pipe and conical surfaces</p> <ul style="list-style-type: none"> ▪ New adaptor for curves defined by analytical ACIS laws (see <code>Aci sLaw_LawEnt</code> class) has been introduced to perform accurate interpolation. ▪ New adaptor is used by a specific approximator class <code>Aci sData_CurveLawBSpl Approx</code> to perform interpolation by means of <code>AdvApprox_ApproxAFuncti on</code> tool. ▪ New adaptive algorithm based on <code>GCPnts_Tangentia lDefl ecti on</code> and actual 3D error returned from the approximator is now used to estimate the maximum number of segments. ▪ <code>Aci sData_MakeCurve: : Bui l dPCurveOfI ntCurve</code> has been modified to re-parameterize P-curves to provide consistent parameterization of the ACIS conical surfaces and the corresponding shapes. |
| <p>22539</p> | <p><i>Summary:</i> Invalid shapes after SAT import</p> <p><code>Aci sData_MakeSurface: : Bui l d</code> method for <code>Aci sGeom_SrfSrfVBI Spl Sur</code> entity has been updated to set the correct parameter range for non-periodic path curves.</p> |
| <p>22569</p> | <p><i>Summary:</i> Invalid faces after SAT import</p> <p>The problem of invalid parameterization of a periodic b-spline surface, produced by rotation of a straight curve has been fixed in <code>Aci sData_MakeSurface</code> class. The problem of invalid parameterization of a seam edge, lying on a periodic b-spline surface, which is produced by rotation of a straight curve, has been fixed in <code>Aci sData_Aci sToCascade</code> class.</p> |
| <p>22606</p> | <p><i>Summary:</i> Invalid shape after SAT import</p> <p><code>Aci sData_MakeCurve</code> has been modified to construct ellipse curves by <code>gp_Ax2</code> and two radiuses, which is more natural than construction of an ellipse by 3 points used previously by ACIS reader.</p> |





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| 22616 | <p><i>Summary:</i> Invalid shapes after SAT import</p> <p>The classes <code>Aci sData_CurveLawAdaptor</code> and <code>Aci sData_HCurveLawAdaptor</code> have been modified to take into account the scale factor of the model during the evaluation of the analytical law used to specify the path curve for translation of "sweep" surfaces.</p> |
| 22617 | <p><i>Summary:</i> Invalid faces after SAT import</p> <p>The problem of invalid parameterization of p-curves generated by demand for a simple face with underlying planar surface and boundaries represented via straight lines has been fixed in <code>Aci sData_Aci sToCascade</code> class by adjusting the normalization process, so as to copy the 3D curve if the validation fails. In this case the problematic edge gets its own 3D-curve representation with a valid parametric range and does not affect other edges having the same 3D curve.</p> |
| 22640 | <p><i>Summary:</i> SAT import: invalid shell comprised by 2 sewed b-spline faces</p> <p><code>Aci sData_Aci sToCascade</code> class has been changed to apply parameterization of underlying b-spline pcurves to seam edges which might come inconsistent from ACIS.</p> |
| 22706 | <p><i>Summary:</i> Exception and invalid shape after SAT translation</p> <p>The class <code>Aci sGeom_Degenerate</code> has been improved to correctly read entities of type 'degenerate_curve' to avoid incorrect mapping of subtypes.</p> |

DXF Import

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| 22174 | <p><i>Summary:</i> Problem with reading of the attached file</p> <p>Method <code>DxfFile_RWLine::ReadField</code> has been modified to allow successfully reading the coordinates arranged in any order, for example: X1,X2,Y1,Y2,Z1,Z2.</p> |
| 22395 22537 | <p><i>Summary:</i> Invalid shape after importing a DXF file</p> <p>An additional shape healing call for wires produced by sweeping via <code>BRepFill_Prism</code> tool has been introduced in <code>DxfData_TranslateObject.cxx</code> to avoid sweeping of broken wires.</p> |

DXF Export

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| 22392 | <p><i>Summary:</i> The attached shape cannot be read by <code>View_3</code> after export to DXF</p> <p>The class <code>Aci sData_Aci sModel</code> has been modified to ensure correct exportation.</p> |
| 22561 | <p><i>Summary:</i> Group code 74 is required in the description of <code>Linetype</code> element</p> <p>Group code 74 has been added in the description of <code>Linetype</code> element in class <code>DxfFile_RWLineTypeElement</code> as required by the current version of DXF format.</p> |





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| <p>22607</p> | <p><i>Summary:</i> Shape imported from DXF has incorrect dimensions</p> <p>DXF reader has been modified to process \$INSUNITS header variable if the interface variable <code>read.dxf.insunits.scale</code> newly introduced in class <code>DxfControl_Controller</code> is set to 1. \$INSUNITS variable can have one of 20 different values identifying the units of measurement to apply (inches, miles, millimeters, etc).</p> <p>If <code>read.dxf.insunits.scale</code> is active, \$INSUNITS variable is used rather than \$MEASUREMENT, which specifies only the measurement system (0 for Imperial, 1 for Metric). If \$INSUNITS variable is not defined in DXF header, the value of \$MEASUREMENT variable is taken instead.</p> <p>Methods for typical & advanced scaling have been added in class <code>DxfControl_ActorRead</code>.</p> |
| <p>22729</p> | <p><i>Summary:</i> State of output stream in DXF write is not controlled</p> <p>The possibility to check the state of the stream and to output an error report has been implemented in <code>DxfFile_FileWriter</code> algorithm.</p> |

Express Mesh

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| <p>22542</p> | <p><i>Summary:</i> Some triangles are out of face boundaries</p> <p>Algorithm of edge tessellation has been improved in <code>QMShape_Tessellator</code> class to take into account the deflection in 2D space.</p> |
| <p>22585</p> | <p><i>Summary:</i> <code>QMShape_Parameters</code> raises exception when processing a shape with void bounding box</p> <p>The class <code>QMShape_Parameters</code> has been protected against getting the dimensions of a void bounding box. In this case the state of the computation will be <code>NotDone</code>.</p> |
| <p>22657</p> | <p><i>Summary:</i> Empty output triangulation on the given face</p> <p><code>Precision::Confusion()</code> has been replaced with <code>Precision::PConfusion()</code> in the triangulation algorithm to avoid losing too small edges.</p> |





Porting to version 6.5.2

Porting of user applications from the previous 6.5.1 OCCT version to version 6.5.2 requires the following issues to be taken into account:

- Any code that has been generated by WOK from CDL generic classes `Tcol I ecti on_DataMap` and `Tcol I ecti on_I ndexedDataMap` needs to be regenerated by WOK to take into account the change in the interface of these classes.
- The enumerations `CDF_StoreStatus` and `CDF_Retri evabl eStatus` have been replaced by the enumerations `PCDM_StoreStatus` and `PCDM_ReaderStatus`. Correspondingly, the methods `Open`, `Save` and `SaveAs` of the class `TDocStd_Appl i cati on` have changed their return value. Any code, which uses these enumerations, needs to be updated.
- `BRepLi b_MakeFace` has been modified to accept tolerance value for resolution of degenerated edges. This tolerance parameter has no default value (to ensure that the client code takes care of passing some meaningful value, not just `Preci si on: : Confusi on`), so some porting overheads are expected.
- If the callback mechanism in `cal l _togl _redraw` function was used in the application code, it is necessary to revise it to take into account the new callback execution and provide a check of reason value of `Aspect_Graphi cCal l backStruct` in callback methods to confirm that the callback code is executed at the right moment. Now the callbacks are executed before redrawing the underlayer, before redrawing the overlayer and at the end of redrawing. The information about the moment when the callback is invoked is provided with the reason value in form of an additional bit flag (`OCC_PRE_REDRAW`, `OCC_PRE_OVERLAY`).
The state of OpenGL changed in callback methods will not be restored automatically, which might lead to unwanted behavior in redrawing procedure.
- The print method used in the application code might need to be revised to take into account the ability to choose between print algorithms: tile and stretch. The stretch algorithm will be selected by default during porting.
- It is recommended to `BRepMesh_Di scretFactory` users, to check `BRepMesh_Di scretFactory: : SetDefaul t()` return value to determine plugin availability / validity.
`BRepMesh_Di scretFactory: : Di scret()` method now returns handle instead of pointer. The code should be updated in the following manner:

```

Handle(BRepMesh_Di scretRoot)
BRepMesh_Di scretFactory: : Get(). Di scret (theShape, theDefl ecti on,
theAngul arTol er);
if (! aMeshAI go. I sNul l ()) {}

```

The default state of `BRepMesh` parallelization has been turned off since 6.5.1 release. The user should switch this flag explicitly:
 - by using methods `BRepMesh_I ncremental Mesh: : SetParal l el (Standard_True)` for each `BRepMesh_I ncremental Mesh` instance before `: : Preform()`;
 - by calling `BRepMesh_I ncremental Mesh: : SetParal l el Defaul t(Standard_True)` when `BRepMesh_Di scretFactory` is used to retrieve the meshing tool (this also affects auto-triangulation in AIS).

If you are porting from an older version of OCCT, check the Porting sections in earlier OCCT Release Notes documents.





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 2008* |
| 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 | <p>GeForce The following versions of GeForce drivers are recommended:</p> <p><i>For Linux:</i> 64-bit Version: 100.14.19 or later 32-bit Version: 100.14.19 or later</p> <p><i>For Windows:</i> Version 266.58 WHQL or later is recommended: http://www.nvidia.com/Download/index.aspx</p> |
| Graphic library | OpenGL |
| C++ | <p><i>For Linux:</i> GNU gcc 4.0. - 4.3.2.</p> <p><i>For Windows:</i> Microsoft Visual Studio .NET 2005 SP1** with all security updates Microsoft Visual Studio .NET 2008 SP1 Microsoft Visual Studio .NET 2010</p> |
| TCL (for testing tools) | <p><i>For Linux:</i> Tcltk 8.5 http://www.tcl.tk/software/tcltk/8.5.html</p> <p><i>For Windows:</i> ActiveTcl 8.5 http://www.activestate.com/activetcl/downloads</p> |
| Qt (for demonstration tools) | Qt 4.6.2 http://qt.nokia.com/downloads |
| Freetype (OCCT Text rendering) | freetype-2.3.7 http://sourceforge.net/projects/freetype/files/ |
| Ftgl (OCCT Text rendering) | ftgl-2.1.2 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 2008 is a permanently tested platform.
- ** The official release of OCCT for Windows contains libraries built with VC++ 2005.
- *** This product is optional.

