

# Open CASCADE 5.2.1 Maintenance Release

## Release Notes

### Overview

Open CASCADE Technology 5.2.1 is a maintenance release, which includes improvements and bug fixes, over previous minor release 5.2.

Version 5.2.1 is binary incompatible with any of the previous versions of Open CASCADE Technology, so applications linked against a previous version (major, minor or maintenance) must be recompiled to run with this Version 5.2.1.

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## Highlights

- Color scale has been implemented in Open CASCADE 3D Viewer.
- The mechanism of system exceptions and signals handling has been improved.
- More convenient use of texture mapped fonts on Windows.
- Performance improvements in mesh visualization services (MeshVS package).

## New Features

### Visualization

- A color scale has been implemented in the OCC 3D Viewer on the basis of `Visual3d_Layer` class. The new class `V3d_ColorScale` class provides possibility to show a color scale typically used in FEA applications right inside the 3d Viewer. This ready-to-use class brings numerous additional benefits to the user providing comprehensive capabilities to customize the color scale by manipulating the following parameters:
  - upper and lower values of data;
  - user-defined number of intervals (colors) in the color scale;
  - possibility to specify a list of colors explicitly (user-defined colors) to override default ones;
  - optional text labels for each interval (color) which would override default labels (numbers) generated automatically for each boundary between two intervals;
  - color scale caption;
  - width and height;
  - position (X and Y coordinates of the top left corner of the color scale in the 3D view window).

### New Products

- The **Express Mesh** algorithm has been introduced. This new feature is not an open source part of Open CASCADE Technology, but a separate product with a commercial license.

The algorithm can be used as a simple quick meshing algorithm. The resulting mesh can be used for visualization instead of standard Open CASCADE Technology meshing algorithm (BRepMesh package), as well as in other applications satisfied by the patch-by-patch meshing solution.

Express Mesh will be available under the site development license (as most other licensed software components at [www.opencascade.org/support/products](http://www.opencascade.org/support/products)). For the initial period it will be available only for current customers with valid support or software license contracts. For details please contact [marketing.contact@opencascade.com](mailto:marketing.contact@opencascade.com).

## Improvements



## Technical documentation

- Mesh Visualization Services description in reference documentation has been completed.



## Foundation Classes

- The mechanism of system exceptions (signals) handling has been improved:
  1. All floating point errors are now converted to OCCT exceptions inheriting the `Standard_NumericError` class.
  2. The function `OSD::SetSignal()` has been enhanced on Windows platform. Now it can work using either the `SetUnhandledExceptionFilter` Win32 API call or the `_set_se_translator` MS Visual C++ runtime library call. The needed method is chosen using the `OSD::UseSETranslator` function.

A large number of floating point errors in various modeling algorithms (extrema, boolean operations, draft features, surface intersector, local and global properties of shapes, shape healing) have arisen after this improvement. They were all fixed, and the reliability and stability of OCCT code in general became much better.

Please take special precautions when migrating to the version 5.2.1 if you used some special handlers for numeric exceptions in your code.

- Compilation warnings reported when using `NCollection` templates have been eliminated on Linux.
- Classes `TCollection_AsciiString` and `TCollection_ExtendedString` use an optimized technique of operations with strings (copy, comparison) by double words (4 bytes). However, previously memory allocation was not aligned on a 4 bytes boundary, which sometimes led (for strings with a length greater than the `MMGT_THRESHOLD` value when `MMGT_OPT=0`) to writing into unallocated memory, resulting in memory corruption (in the form of exceptions or debug asserts). Now the memory allocation in these classes is aligned on a double words boundary (where necessary) so that memory corruption is excluded.
- Copy constructor of the `Standard_Transient` class has been corrected to initialize field count by 0, thus ensuring a correct value of this field when classes inheriting `Standard_Transient` are copied.
- Open CASCADE memory manager (its function `Standard::Allocate()`) has been fixed to work correctly with free lists in case when `MMGT_OPT=1` and `MMGT_CLEAR=0` (memory losses and incomplete copying in method `Standard::Reallocate()` are avoided).





### ✓ Modeling Data

- Class BRepAdaptor\_CompCurve has been optimized due a bug fix which eliminates the segmentation fault when the Edge() method is used.

### ✓ Modeling Algorithms

- The BRepAlgoAPI\_Section algorithm now does not fail in cases when unacceptable small edges are present in the shape but situated at a long distance from the section plane. Earlier such edges were obstacles to perform an operation regardless of how long the distance from them to the area of intersection is. This improvement allows to perform section for a valid part of the model, while section for an invalid part will still be impossible (according to Boolean Operations limitations).
- The API of the BRepPrimAPI\_MakePrism algorithm has been improved: an optional argument has been added to the constructor of this class to disable surface canonization. It is useful when we expect U-isolines of built surfaces going along original curves while V-isolines being parallel to the direction of extrusion.
- The low level internal algorithm responsible for searching a rough intersection between a curve and a surface and used in Boolean operations has been improved.
- The section part of Boolean operations has been improved: for some cases the section between shapes is tested to reject produced edges that are too small (with only one vertex completely covering the edges). Now the result of section is valid for further processing.
- Intersection of a surface of linear extrusion with a plane parallel to the direction of extrusion now stably produces a direct line instead of a set of overlapped B-spline curves.
- Some errors in internal root algorithms for the 'sprops' (surface properties) computing have been fixed.

### ✓ Visualization

- Several cases of uninitialized fields in Visualization and OCAF classes have been detected and corrected, to avoid problems when Open CASCADE Technology applications run with environment variable *MMGT\_CLEAR* set to 0.

### ✓ Application Framework

- Drivers for persistence (OCAF, Xml, and Binary formats) have been implemented for OCAF attribute TDataStd\_ExtStringArray.
- LDOMParser now correctly treats the DOCTYPE declaration with external ID, i.e. containing the URI of a DTD file. As before, this DOCTYPE declaration is ignored in the parsed document and cannot be passed to the XML output by a subsequent call to LDOM\_XmlWriter.



### Data Exchange

- StlAPI\_Reader (translator) has been corrected to apply the sewing algorithm on all faces at once (all faces are collected in one compound, then the sewing algorithm is applied) instead of sewing faces one by one.

### Test Harness Draw

- The problem with presentation of some faces built on periodical surfaces in the DRAW application has been fixed.

### Samples

- **Visualization Performance Method** sample:
  - Some functionality has been implemented that was declared in sample dialogs but actually didn't work;
  - "Stop automatic testing" functionality has been added;
  - Enhanced sample look'n'feel (dialogues);
  - Some minor bugs have been fixed.

### Building Tools

- The procedure of compilation of user applications built on the basis of OCCT binary distribution package has been simplified:
  1. The specification of --with-tcl, --with-tk arguments of the configure script are no longer mandatory. The configure script will search for tclConfig.sh/tkConfig.sh files in default locations, such as /usr/lib, /usr/local/lib for Linux, /opt/sfw/lib, /usr/tcltk/lib/itc for Sun Solaris. Even if location of tcl/tk installation isn't found by configure script, it will continue working.
  2. In order to exclude the possibility of collisions between the \$CASROOT/inc/config.h file supplied with binary distribution and the \$CASROOT/config.h file generated by configure script, a step of creation of a symbolic link between \$CASROOT/inc/config.h and \$CASROOT/config.h has been added to the configure script, so if the user executes configuration the \$CASROOT/inc/config.h file supplied with binary distribution will be replaced by the generated one.





## Changes

### Visualization

- The performance bottleneck has been fixed by using `Select3D_SensitiveGroup::Add` method with a list as an argument. The old version used method `Add()` with one sensitive entity as an argument. This variant of method works slowly because every time it runs through the list looking for an entity. Now the user can create a full list and pass it to `Select3D_SensitiveGroup`.
- `MeshVS_Mesh::ComputeSelection`:  
Previously the default selection mode was 0. Now this mode is intended for selection of the whole mesh. The default selection must be of any value different from `MeshVS_EntityType` flags and zero.
- The behavior of the "meshselmode" DRAW command has changed as follows:
  - 1) if selection mode = -1, the command closes all local contexts, therefore selection is impossible;
  - 2) if selection mode = 0, only 0 selection mode (whole mesh) is possible;
  - 3) if selection mode > 0, the 0 selection mode becomes inactive (this solution fixes the bug with slow selection).
- `CSF_TEX_FONT` environment variable has been removed. Currently bitmap fonts are used by default.

To switch texture mapped fonts on / off:

`Graphic3d_AspectText3d::SetTextureMappedFont(Standard_Boolean)` method should be used with the `Standard_True` / `Standard_False` argument. First, a `Graphic3d_AspectText3d` object should be prepared and then this object can be used in a usual manner, i.e. passed to the `Graphic3d_Group::SetPrimitivesAspect()` method.

Without explicit calls to `Graphic3d_AspectText3d::SetTexturedMappedFont()` the objects are created using texture mapped fonts.

- Some class names have been changed. The following table describes the correspondence between old names of classes instantiated in the MeshVS package of the previous versions and new names of the current version (5.2.1):

Previous version (5.2)	Current version (5.2.1)
MapOfColor	DataMapOfIntegerColor
MapOfMaterial	DataMapOfIntegerMaterial
MapOfInteger	DataMapOfIntegerBoolean
MapOfOwner	DataMapOfIntegerMeshOwner
MapOfText	DataMapOfIntegerAsciiString
MapOfTwoColors	DataMapOfIntegerTwoColors
MapOfVector	DataMapOfIntegerVector
TwoColorsToIdsMap	DataMapOfTwoColorsMapOfInteger
ColorToIdsMap	DataMapOfColorMapOfInteger

The names of iterators have been changed in accordance with Open CASCADE Technology rules, for example:



For `DataMapOfIntegerVector` the iterator now has the following name: `DataMapIteratorOfDataMapOfIntegerVector` and etc.

Besides:

1. Methods in the `MeshVS_Drawer` now take the `MeshVS_DrawerAttribute` enumeration, not the `Integer` one as they did earlier (however this approach has a serious drawback, bug 7088).
2. In the constructor of `MeshVS_PrsBuilder` and its successors the first parameter is `Handle` to `MeshVS_Mesh`, but the builder contains a pointer to `MeshVS_Mesh` as earlier and in constructor it extracts pointer from `handle`.
3. The new `MeshVS_DrawerAttribute` enumeration item has been added: `DA_ComputeSelectionTime`, if its value is true, a string with information about the run time of the `MeshVS_Mesh::ComputeSelection` method will be passed the standard stream of output.

### ✓ Application Framework

- Class `BinMDataStd_ExtStringArrayDriver` has been added to package `BinMDataStd`.
- In the constructor of `LDM_OSStream` class, call of the `Init(streambuf*)` method of the base class has been added. This prevents incorrect initialization of base class fields on some platforms.

### ✓ Data Exchange

- Full circles are now translated into a circle IGES entity instead of NURB entity in the past.

### ✓ Porting

- Open CASCADE Technology has been ported to Solaris 2.8 with the support of Forte 6.2 compiler. This configuration is not currently planned to be officially supported in the future releases. It has been provided within a customer project and may only signal that version 5.2.1 works on that configuration with the same quality as on other officially supported configurations. Binaries delivered with this Version 5.2.1 of Open CASCADE Technology have been built with Sun Solaris 2.6 and Sun WS 4.2 compiler.

## Bug Fixes



- Open CASCADE 5.2.1 incorporates **58** modifications (bug fixes, enhancements and other corrections) over version 5.2. For details, refer to [Appendix 1](#).



## Appendix 1: Open CASCADE 5.2.1 Bug Fixes

- [Foundation Classes](#)
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Foundation Classes, 8 bug fixes	
ID	Short Description
984	LDOM hangs-up attempting to read a file containing a reference to a .dtd-file.
6143	Corrected handling of exceptions on WinNT.
6184	Compilation warnings when using NCollection templates.
6583	Malloc/free problem in NCollection_Vector.
6650	Problems with SALOME compilation on CASCADE 5.2 installed from CD to Linux RH8.
6794	AsciiString corrupts memory if MMGT_OPT is equal to 0.
6804	Class Standard_Transient has copy constructor which does not initialize field count.
6865	Standard::Allocate works incorrectly if MMGT_OPT=1 and MMGT_CLEAR=0
Modeling Data, 2 bug fixes	
ID	Short Description
5696	Exception in BRepAdaptor_CompCurve due to uninitialised variable CurIndex.
6503	Incorrect calculation of bounding box for infinite surfaces
Modeling Algorithms, 20 bug fixes	
ID	Short Description
5886	An optional argument is added in the BRepPrimAPI_MakePrism class constructor to disable the simplification of linear extrusion surfaces to planes.
6063	Exception is raised during command "spros" with epsilon <1 on attached shape in DRAW.
6272	Incorrect results of gluing operation.
6181	Incorrect result of boolean cut
6182	Incorrect result of boolean cut
6277	Cut operation does not produce any result.
6278	Fuse operation produces incorrect result
6279	Incorrect section between two solids
6281	Cut operation is invalid (caused by wrong section between arguments, it contains a new edge instead of the edge from one of the arguments.)
6282	Cut operation is invalid (caused by wrong section between arguments, it contains edges with one vertex, but these edges are neither degenerated not based on closed in 3D or 2D curve.

6289	Internal error (code 101) in Section computation.
6334	Invalid result of MakeCylindricalHole when input solid is reversed.
6371	Performance of some classes improved.
6390	ExpressMesh algorithm has been integrated as a separate product.
6450	Bcommon operation does not work with operand halfspaces made of two parallel infinite planar faces, and having a common space between these faces.
6474	Intersection of a plane and an extrusion surface gives incorrect result (4 overlapping bspline curves instead of 1 direct line).
6502	Wrong calculation of ShrunkRange algorithm.
6602	New Boolean operations return error on Solaris 2.8 with Forte 6.2
6605	Exception at offset operation on Solaris 2.8 with Forte 6.2
6615	Some sources are modified to introduce a special algorithm of partition.
<b>Visualization, 6 bug fixes</b>	
ID	Short Description
5682	AIS_InteractiveContext::Remove does not remove the object from memory.
6247	Color scale in OCC 3D Viewer.
6496	Performance problem of loading big STL shape by meshfromstl DRAW command.
6571	Bitmap and Textured mapped fonts optimization.
6652	Problem of mesh selection.
6802	Exceptions in visualization when MMGT_OPT=0 and MMGT_CLEAR=0
<b>Application Framework, 3 bug fixes</b>	
ID	Short Description
6340	BinOcaf reader is now compatible with LH3D binary files.
6493	Binary persistence is included in the TDataStd_ExtStringArray standard attribute.
6604	Detected and improved some exceptions at reading files in the Xml format on Solaris 2.8 with Forte 6.2.
<b>Shape Healing, 3 bug fixes</b>	
ID	Short Description
5215	Face #1923 can't be split in two faces.
5439	A wire from a face can't be split in ShapeFix_Wire. See also bugs 55 and 5215.
5783	Logical error in ShapeFix_TransferParametersProj::TransferRange.
6805	Operation BSplineRestriction produces shape with null p-curves.
<b>Data Exchange, 12 bug fixes</b>	
ID	Short Description
55	Invalid shape is obtained after reading from a STEP file.
5079	Exception in the translation of IGES files.
5099	Invalid shape is obtained after the translation of attached file.
5953	Wrong STEPCAFControl_Reader constructors.
6283	Unstable work of fixshape on attached shape (different number of warnings).
6284	Exception while reading the file attached to bug 5708. See also bug 5708.
6384	StlAPI_Reader still uses old Sewing algorithm
6491	After sewing in DRAW of the attached shape with tolerance 1000, the resulting shape





	contains duplicate faces (277 instead of 263).
6508	Reading invalid General Symbol entity from an IGES file causes an exception.
6542	Circle is translated to IGES as NURBS.
6675	Sewing does not work for attached shapes with any tolerance. See bug 6542.
6805	Operation BSplineRestriction produced a shape with null pcurves.
<b>Test Harness - Draw, 2 bug fixes</b>	
ID	Short Description
5809	DPLUGIN macro gives a compilation warning on Linux
6132	Wrong ISOS presentation of faces built on a periodic surface in Draw.
<b>WOK, 2 bug fixes</b>	
ID	Short Description
5916	The attempts of WOK to compensate time displacement between current workstation and server sometimes make it impossible to work.
6786	WOK does not close unused files, and it leads to errors when a workbench is deleted.

