CAD | BIM | MECHANICAL

DWG
MECHANICAL SDK

SIG

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OVERVIEW

ODA Mechanical SDK is a vertical extension based on Drawings SDK to access DWG mechanical objects stored in Autodesk® Mechanical 3D® files.

- The following Mechanical data standards are supported: ANSI, BSI, CSN, DIN, GB, ISO, JIS

- Full support of basic functionality for 22 of 28 2D Mechanical entities is provided:

  - Visualization

  - Example code of entities creation

  - Read and write access to entities properties via SDK API

  - We are going to support the StdPart entity

- Automatically generated Bill Of Materials and Part Lists based on Mechanical structure data are supported

- Example applications for all supported Mechanical features are available

WHAT'S NEW 2022

- Implemented saving of Mechanical database to Autodesk® AutoCAD® .dwg file format. Mechanical objects converted to blocks and saved to a Autodesk® AutoCAD® .dwg file. dxf format is also supported.

- Extended the public API (added new mechanical API methods and AMI functions).

- Copy-paste functionality enhancements.

  Two directions can be distinguished here: this is copying using deepClone() and using wblockClone(). Last method used for copying objects to an existing database and to a new one and used by "clipboard"(ctr+c). In both cases, the copying mechanism was improved, considering the features of other objects attached to the copied object, as well as considering the type of operation performed. Also 'inset' operation was improved.

- Numerous improvements and fixes were made to BOM functionality: the AcmBOMManager class was improved to correctly handle items from attached BOM tables; functionality of the AcmDataEntry class was improved to work correctly with xref data entries. Improvements were also made to handling of external component definitions. External component definitions can now be reliably detected by the amiIsCompDefExternal() function.

- The code responsible for expanding/collapsing rows in the BOM table was also refactored and the functionality of the corresponding methods (expand/collapse) was improved, as a result of which it became possible to collapse/expand rows of information in which information is located in external files (Xref).

- The AcmCPartList class was improved to work correctly with empty Parts Lists and Parts Lists with excluded items, correctly handle text wrapping in headers and fix the number of issues in rendering.

- Functionality of the welding symbol has been improved. Welding services (subsymbols of the welding symbol) are arranged and aligned for the following standards: ANSI, DIN, ISO, JIS. Welding service properties have been fixed for the correct operation of the AcmWeldingStd::getWeldDesc() method. Welding service names are corrected for ANSI, DIN, ISO and JIS standards to be compatible with Autodesk Mechanical. AcmCWeldService::serviceName() and AcmCSymbolDesc::description() methods return correct values now.

- The layer configuration manager functionality has been improved so that layer configurations can be modified (to change a layer) and deleted.