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NEW WAYS TO CONFIGURE THE PARAMETRIZED MODEL DOCUMENTATION

Abstract: Manufacturers are moving toward 3D printing and CNC machining and 2D documentation, with traditional dimensioning and tollerancing becoming less relevant. MBD allows the addition of the dimensions and surface finishes that would have been traditionally communicated via 2D drawings. Existing electronic formats and viewers provided by most of the large commercial CAD-CAM products and electronic signatures legislation exclude, in most cases, the necessity of hard copy printing and the project data base is easely handled using the BOM structures. This papers will present the configuration of the model based definition and solution for linking the design knowledge content, using Autodesk Inventor®.

Key words: model based definition configuration, CAD, Inventor

1. INTRODUCTION

Deriving the operational data for manufacturing parts with CNCs is common practice and G codes are directly derived from the geometry of the 3D model. When human operators handle the manufacturing operations and assembling, thorough bidimensional documentation, using standard annotations is mandatory. Existing electronic formats and viewers provided by most of the large commercial CAD-CAM products and electronic signatures legislation exclude, in most cases, the necessity of hard copy printing and the project data base is easely handled using the BOM structures.

Next obvious step is to eliminate the 2D standard layout representation of the model, with projections and sections displayed and annotated on the virtual paper. Model based definition offers a fairly solution to the model documentation, an intermediate between the CNC method of apraisal of the model data and the need to explicitely expose the geometry and technological detalis to a human operator.

This new CAD design technology is allready in use in automotive, aerospace, heavy equipment industries, military and defence sectors as well as by some specific product manufacturers.

Capturing the Design intent is critical in any 3D construction and any layout should be focusing on the most relevant structure of the drawing annotations. model based definition is offered as a tool in many modern CAD software and its configuration to suit the best one's design intent brings at ease the model documentation.

This paper will present the configuration of the model based definition using Autodesk Inventor® (MBD-3D model that includes associative product and manufacturing information that defines the product in a manner that can be used effectively without a drawing, according to ASMEY14.41-2012). First, the configuration of the active or unused standards can be modified. The paper offers an example of best practice for a thorough management of the associative defintions styles in the Styles and Standard Editor.
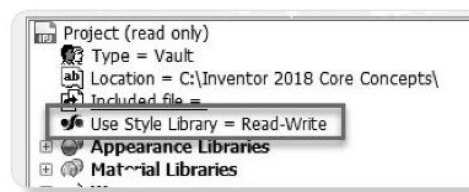

Fig. 1 Project panel for modifying a library content

A case study is presented for the model based definition using the Tolerancing tools and how this data can be retrieved for further layout representation or linked from outside Inventor, if required.

2. DESIGN KNOWLEDGE CONTENT

A digital prototype should support the various types of information for downstream manufacturing and assembly. Jihong & Zhao (2013) described this information as functional, mathematical (constraints, formulas, statistics), engineering (tolerancing, surface conditions, material) and manufacturing (process, equipment, assembly) knowledge [1]. The challenge is to organize and implement a ruling interactive information exchange between the knowledge system and the integration system.

Geometric Dimensioning and Tolerancing (GDT) accompanies the entire product design and development process chain and is a multifold concept: is a language which describes a part based on drawing, consists of a library of symbols, rules, definitions and concentions to describe a part, based on size, form, orientation, location and tolerances, describes the function of a part or assembly, using tolerances [2].

Concretly, the function of a part is defined if are defined its size, form, location and orientation [3].

During CNC production, the planned CAD shape is generated, then verified through measurements, the point cloud extracted is compared with the nominal, ideal planned form. Depending on the results of this final association step, the tool path is permanently adjusted via the active Numerical Control.

For the size component, the functional purpose of a feature's size can be captured directly on the solid model using a specific modifier that captures the mathematical background of the design intent. The standards for ISO 1101 and ASME Y 14.5 differ only at the nevel of detail in the computation method for the geometrical deviation, as implemented in some CAD systems [4].

In 2D, this will need supplementary standardized symbols, to be near the ideal measured value.

The 3D model heavily carries the ideal geometrical definitions, but, for integrated engineering of the product, a deeper technological content is needed. Ussualy this content is explained in several addenda to the geometrical descriptions, as notes, tables on the layout or annexed to the drawing. In order to create an inspection plan, ussualy, along with the common 3D CAD model a blueprint with datum and tolerances is needed [5, 6]. An augumented model could transfer all the inspection data to an inspection program, with dependencies automatically updatable, which will save testing timp and reduce the inherent human errors [2, 7]. One of the main issues of migration to the new 3D modeling approach is the selection and prioritization of the legacy data for the transition.

The product development is impacted troughout its stages by the CAD, and transfering to a model based definition will improve not only the communication with tenders (suppliers, manufacturers, clients), but will impact on the conceptual, system design and production planning, as well. Choosing MBD means to understand the difference between a part presentation (layout, paper) and a representation.

MBD consists of the geometry (geometrical support defined by the basic dimensions), the annotations (notes, dimensions, tolerances, and characteristics), attributes (parameters, custom properties, functions, in general meta and queried data) and presentation (views, cameras, capture, combination states, presentation organization). One can identify the main difference between MBD: while the draqings require all basic dimensions and still remains ambiguos, the MBD is the basic dimension, with absolute representation precision [8].

Annotations are semantic, digitally associated to the features, pattern and enforcing a standard. Here, the tolerances can reffer to the overall profile tolerance, or specific geometric feature tolerance. Few systems automate the collection of surfaces for overall profile tolerance.

The ways of design knowledge differ structuraly from the production knowledge. While the design knowlege emerges as a capitalisation of the every day knowledge, it is prior and episodic knowledge is strongly emotional, subjective and tacit and has declared objective bases, the production knowledge is rooted in specific combinations of prior knowledge, directly linked with the machine and production lines experience.

3. ORGANIZING THE 3D ANNOTATIONS

Using design views, the content of the Annotation folder can be displayed in each of the views, making visible only the model annotations that apply to the respective view, as each of the views remembers the status of the annotations within.

The model space browser in Inventor contains the Annotation folder, which stores the 3D annotations added to the model.
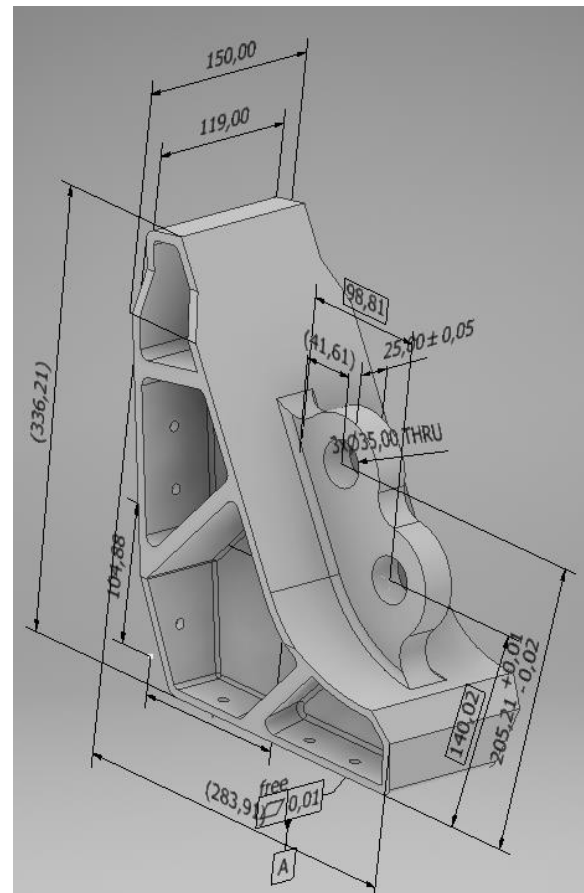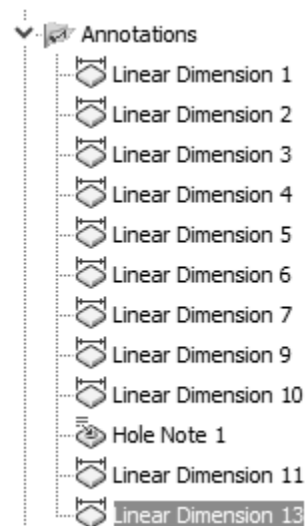

Fig. 2 The 3D model in Iso View

Fig. 3 Annotation folder excerpt

The common practice of ortographic Design Views is now enhances with the ability to display automatically the 3D annotations, with the possibility to hide some, in specific views, where they do not apply. The component

annotations and 3D annotation visibility can be tunned also from the View Tab, using the Object Visibility Command.
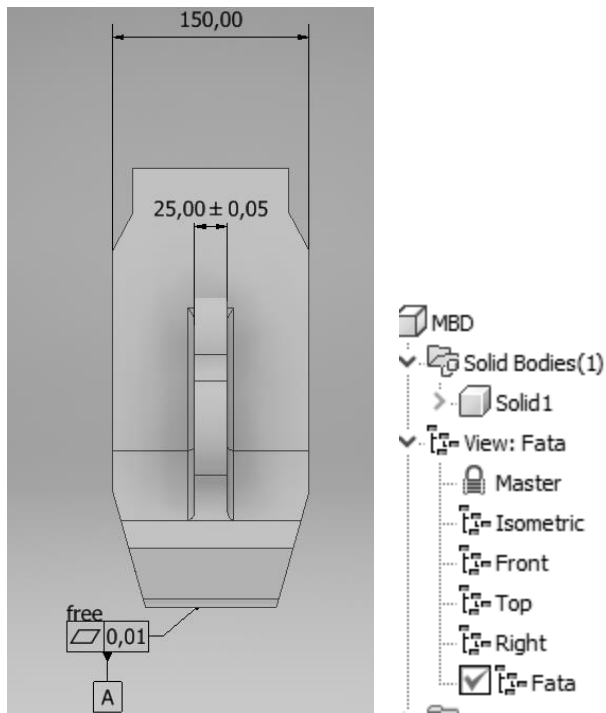

Fig. 4 Selected annotations, filtered by the Design View

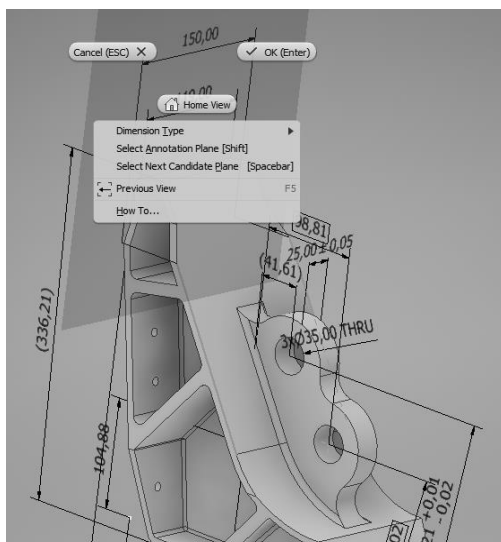In figure 5 is presented the model with the plane selection, while dimension is defined within a Design view.

Fig. 5 Plane selection for dimensions

A usefull application of the newly integrated annotation methods applies to the assembly. Even though the technical drawing instruments are valid or have sense in the part fabrication instructions, general notes, holes data, surface daat, dimensions can be added an be accessed from the assembly, as well.

4. LINKING FURTHER THE DESIGN KNOWLEDGE

A strong option that successfuly replaces the 2D representation is the 3D PDF export. Important metadata can be exported, together with the geometric description and visualization options, via the structured Design Views.
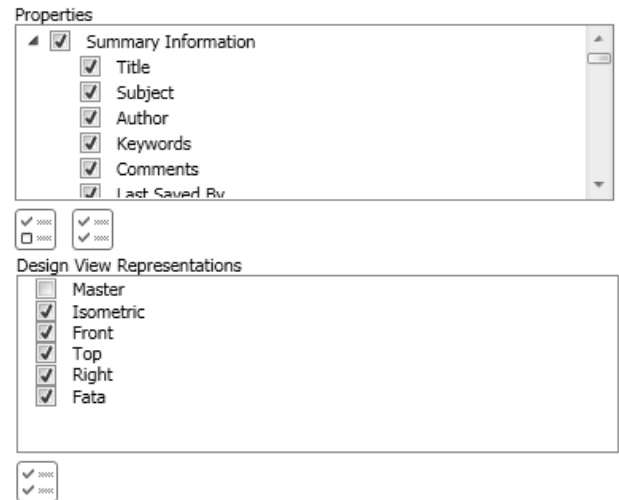

Fig. 6 Export options for 3D PDF format

One very important aspect of the MBD data management is the export options. STEP format is one of the most accesed exporting formats for manufacturing and migrations to other CAD software. In Inventor, the most important STEP format export option is the application protocol 242, for the management of the mode based 3D Engineering.

MBD emphasise the role of the design parameters. The Parameters manager retains all the model defined dimensions, while offering the possibility to define new ones, to be directly related to the model defined with a hierarchical logic. The custom defined parameters can become independent, in raport with the existing ones, obtaining the first basic structure for an iLogic future definition of similar, variants of the same part.

Fig. 7 Excerpt of the defined parameters list

An interesting application for the parameters management is the so called dwg underlay technique, which permits to create a live link to AutoCAD data, in planar representation, up to the 3D Model. In Inventor, dwg formats can be imported, together with their block (and corresponding nesting levels) and layer definitions. When this happens in an iam file, the Make Component command allows transforming the 2D block in an independent part that can be easily contrained in a planar sketch of the future assembly, somehow following the concept of skeleton design. What happens with the parameters defined in the AutoCAD file, at 2D level?

They will be imported as well as usable basis for further design.

The planar drawing imported in an assembly file, correctly aligned with the relevant components, can be placed relatively, using the Locate function, and used as basic sketch for the future new component of the assembly. The 3D development functions, such as extruzion, can benefit from the concept of reference dimension, extracted as such from the orgonal views represented on the dwg.

The most important advantage of this method is that the sketch used to develop new components has a live link with the AutoCAD dwg source. Since AutoCAD still is the design environment for many, it is very important to make use of this facility, as the only secure method to maintain up-to-date parameters in the assembly. Working with external sketches can be handled up to the level of model based definition, using the Parameter Manager facilities, on the hyerarchical scheme dwg - > ipt - >iam.

While exploring the possibilities to work with the parameters, the possibility of copying the design emerges from on the less used application related to Inventor: Design Assistant. While, in Inventor, this option is reserved only for Vault project people, the Design Assistant offers the facility to define the assembly component status as Work in progress.

Copying the design means to take an existing model, making a copy of it with the possibility to change some of its parts, avoiding the design from scratch work. Bassically, some of the part can be swaped during the copy, obtaining new assemblies. Using Parameters Manager and common redesign techniques, the MBD can migrate and evolve form a design to another, without being forced to work under the Vault project principles.

In order to achieve this, the Copy Action needs to be selected (figure 8) in the assembly name file, while giving a new name for the future copy. This should be done with each of the new components of the future assembly, i.e. the components that will be edited in new version of the assemblies. After completing the Copy-Action and Renaming the parts, the assmebly can be saved and in the project tree it will be visible with all the structure, parameters copied and renamed parts ready to be edited as required in the new design concept.
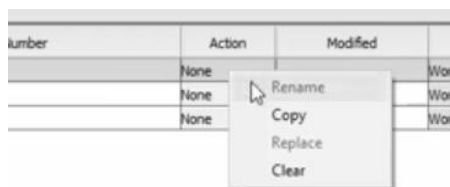

Fig. 8. Design Asistant operation for copy design

5. CONCLUSIONS

Model based definition is a versatile solution to transver unambigously the model data to the fabrication or further CAD operations. The parameters obtained directly dimensioning and tollerancing the model are accesible via Parameter Manger and can be exported as metadata in 3D PDF formats.

More, the design parameters can be obtained from dwg layouts, refferencing the imported dimensions to the 3D operations, thus obtaining a live link with an external sketch, while keeping the unitary design data structure.

MBD structures various types of information for downstream manufacturing and assembly, i.e. associative product and manufacturing information that defines the product in a manner which can be used effectively without a drawing

Design views can be used as visualization filter, both in model and layout, if 2D documentation is required for the model.

REFERENCES

[1] Jihong Liu, Zhenjie Zhao, (2013), *A Study on 3D Model Based Knowledge, 10th International Conference on Fuzzy Systems and Knowledge Discovery*, ISSN 78-1-4673-5253-6/13, pp. 498-500, IEEE.

[2] ***, *Model Based Definition Promises Big Dividends for Quality assurance*, available at *https://www.hexagonmi.com/solutions/technical-resources/technical-articles/model-based-definition-promises-big-dividends-for-quality-assurance, Accessed* 2018, 7 Nov.

[3] Dick Bourke, (2016), *How Model-Based Definition Can Fix Your CAD Models*, available at *https://www.engineering.com/DesignSoftware/DesignSoftwareArticles/ArticleID/11909/How-Model-Based-Definition-Can-Fix-Your-CAD-Models.aspx?e_src=relart* Accessed 2018, 7 Nov.

[4] G. Alducin-Quinterro, (2011), *Social annotation as a tool for dynamic capture process of CAD design intent in the context of knowledge based engineering,* Ph.D. Thesis, Univ. Tecnologico de Monterrey, 2011

[5] Shawn P. Ruemler, Kyle E. Zimmerman, Nathan W. Hartman, Thomas Hedberg, Jr. and Allison Barnard Feeny, (2016), *Promoting Model-Based Definition to Establish a Complete Product Definition*, J. Manuf. Sci. Eng 139(5), 051008 (Nov 21, 2016), Paper No: MANU-16-1446; doi: 10.1115/1.4034625.

[6] ***, *About 3D Annotation and Model-Based Definition*, available at *https://knowledge.autodesk.com/support/inventor-products/learn-explore/caas*, *Accesed* 2018, 28 October.

[7] Herron Jennifer B., (2013), Re-Use Your CAD: *The Model-Based CAD Handbook*, Createspace Independent Pub, ISBN 1494877171, New York.

[8] Rainey Larry B., Tolk Andreas, (2015), *Modeling and Simulation Support for System of Systems Engineering Applications,* John Wiley & Sons, ISBN: 978-1-118-46031-3, New York

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