SOLIDWORKS INSPECTION
WHAT'S NEW 2018

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SOLIDWORKS INSPECTION – ADD-IN

SOLIDWORKS PART AND ASSEMBLY SUPPORT

Users can create complete inspection reports using SOLIDWORKS part and assembly files (*.sldprt, *.sldasm). Start by opening a SOLIDWORKS file.
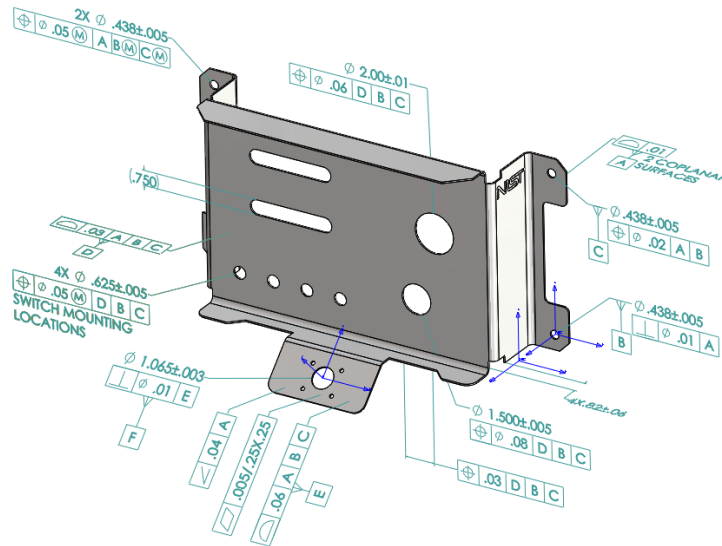

If the part or assembly contains 3D annotations or Product and Manufacturing Information (PMIs) then users can create a new inspection project and extract all the information they need for their inspection checklist.

The process is similar to working with 2D drawings (*.slddrw). Users first click on "**New Inspection Project**"

 from the CommandManager and then define the project settings and extraction settings.

Using simple checkboxes they can setup their project and include or exclude Dimensions, Notes, GD&Ts, Hole Callouts, etc… Then, SOLIDWORKS Inspection automatically adds the balloons to the PMIs so that quality inspectors can refer to the inspection spreadsheet and the 3D model to understand the characteristics to inspect.
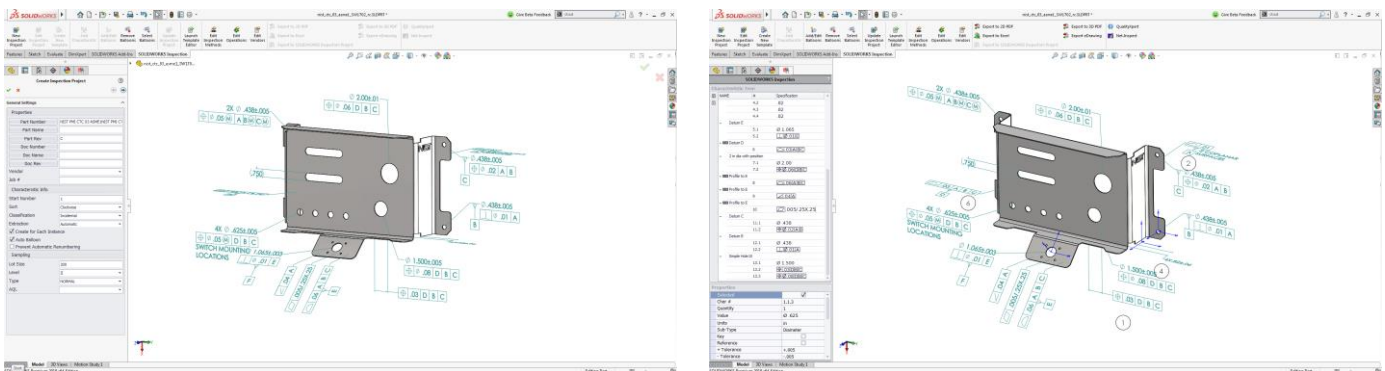
In addition to the Automatic extraction mode, the Manual mode is also available.

Similar to working with a SOLIDWORKS drawing, the characteristics are listed in the Characteristic Tree and users can modify each characteristic's properties to include additional information such as the Operation, Classification, Method of inspection, or just re-order them.
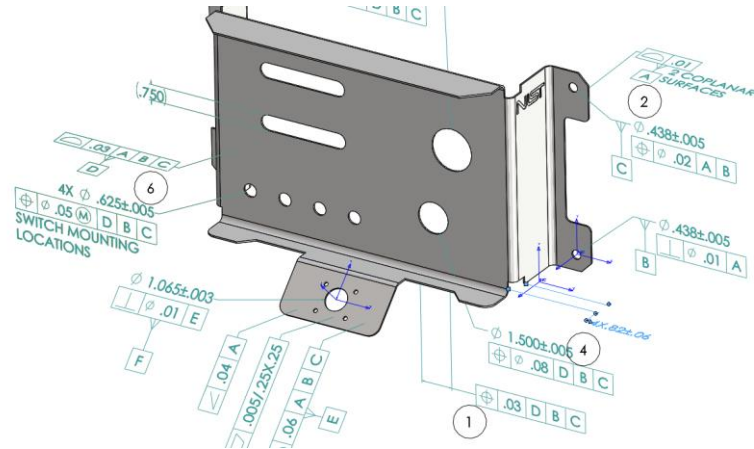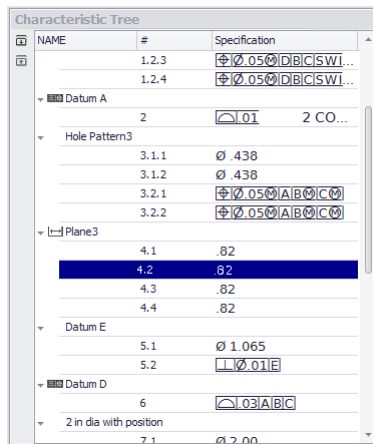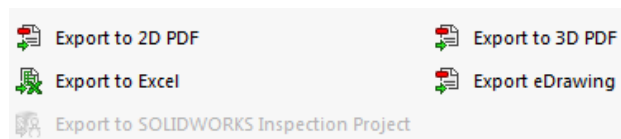

Once the inspection project is completed, users can generate a Microsoft ® Excel report, a 2D PDF, a 3D PDF (if SOLIDWORKS MBD is available) or an eDrawing file.

Benefits: By leveraging existing 3D SOLIDWORKS files users can streamline their quality control and documentation processes. The support of SOLIDWORKS parts and assemblies also enables companies to expand their drawing-less manufacturing strategy all the way to the quality department.

PDM INTEGRATION

SOLIDWORKS PDM integration is now part of SOLIDWORKS Inspection Addin 2018. This new integration helps quality engineers manage and centralize the storage of their inspection projects as well as all the files and reports related to those.

Benefits: Thanks to the SOLIDWORKS PDM integration, users can manage their overall product development with ease and include quality management as an integral part of the process.
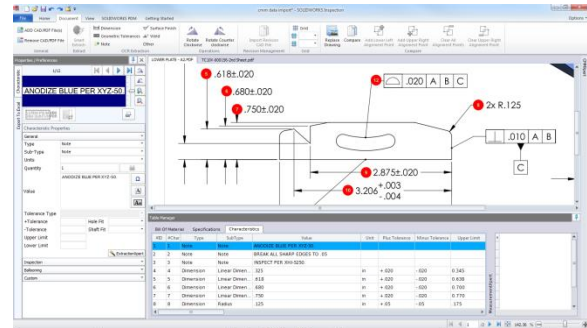
SOLIDWORKS INSPECTION – STANDALONE

IMPROVED USER INTERFACE

SOLIDWORKS Inspection Standalone 2018 user interface is improved for added clarity. Menus, dialogs, buttons are more modern and more consistent across the software. Options and settings are also reorganized. True and False status (for Key characteristics, Basic characteristics, etc.) is displayed in the Bill of Characteristics as opposed to a checkmark.
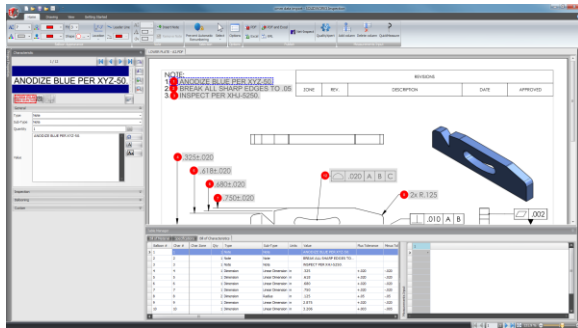

SOLIDWORKS Inspection 2017

SOLIDWORKS Inspection 2018

Benefits: Cleaner and more consistent user interface (UI) for improved productivity.

3D FILES SUPPORT

Users can open and import 3D files from other CAD vendors directly within their inspection project. The following formats are supported with the 2018 release of SOLIDWORKS Inspection:

- 3D XML files
- PTC® and Creo Parametric files
- CATIA® V5 CATPart and CATProduct files
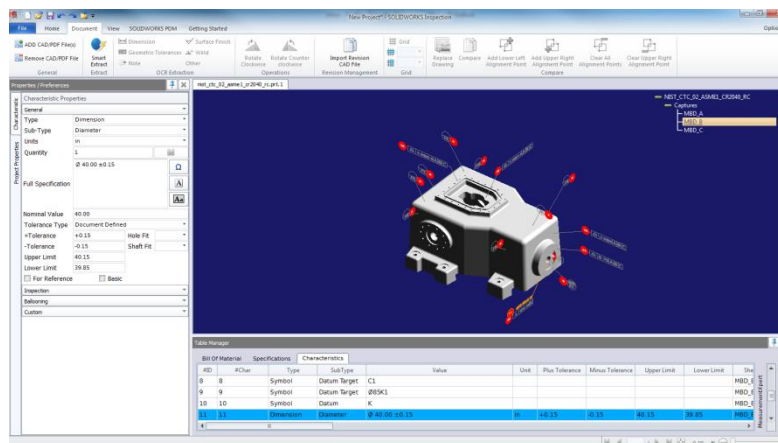
Once the files are imported in the project, the 3D Product and Manufacturing Information (PMIs) can be extracted to create an inspection report.

Characteristics can be simply extracted using the new Smart Extract feature that directly read the 3D file information based on pre-defined settings:
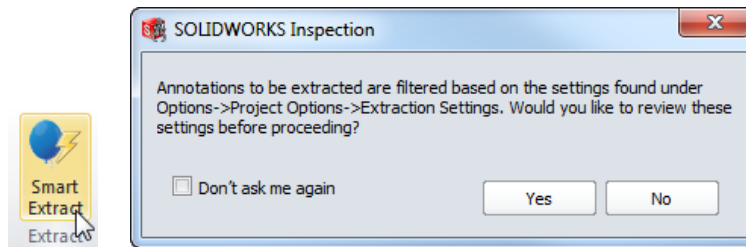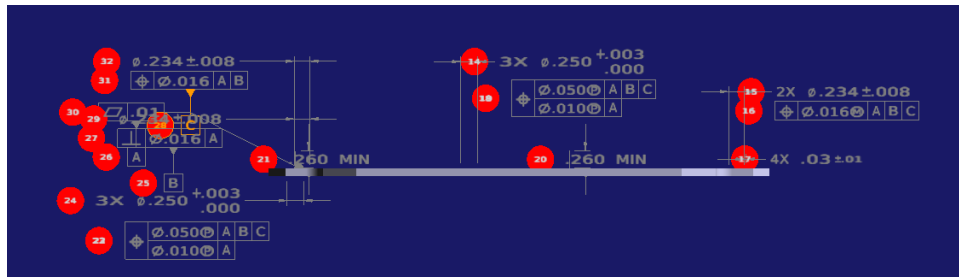

Using those extraction settings, users can automatically include or exclude characteristics such as dimensions, notes, GD&T, etc.
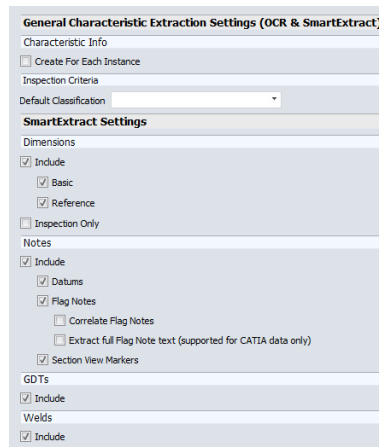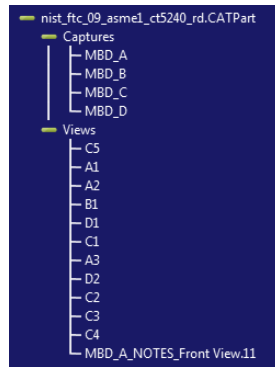

Once the extraction settings are specified, users can click on individual characteristics to add them to their report or can box select multiple characteristics.
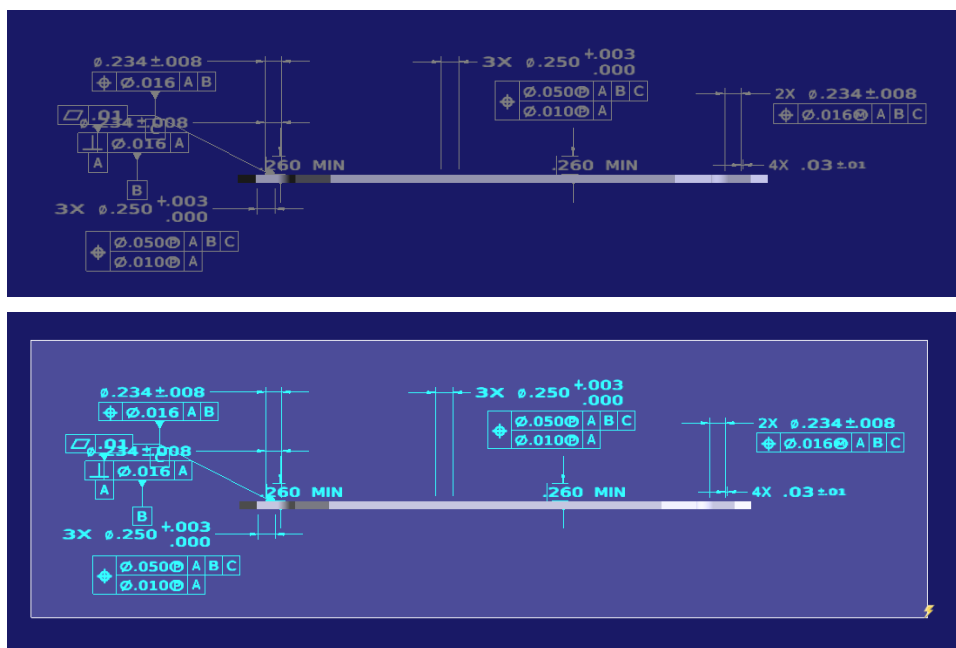
Users can navigate between the different views of the 3D file using the navigation tree in the top right corner.

Similar to working with PDF or TIFF files, the characteristics are ballooned and added to the Bill of Characteristics.

#ID	#Char	Type	SubType	Value	Unit	Plus Tolerance	Minus Tolerance	Upper Limit	Lower Limit	Sheet/View/Capture	Opera
21	21	Dimension	Linear Dimen...	.260 MIN	in				.260	A1	
22	22	Geometric To...	Position	⊕ Ø.050Ⓜ A B C	in			.050		A1	
23	23	Geometric To...	Position	⊕ Ø.010Ⓜ A	in			.010		A1	
24	24	Dimension	Diameter	3X ø .250 +.003/+.000	in	+.003	+.000	.253	.250	A1	
25	25	Symbol	Datum	B						A1	
26	26	Symbol	Datum	A						A1	
27	27	Geometric To...	Perpendicula...	⊥ Ø.016 A	in			.016		A1	
28	28	Symbol	Datum	C						A1	

Once the inspection project is completed, users can generate a Microsoft ® Excel file, a 2D PDF or a 3D PDF.
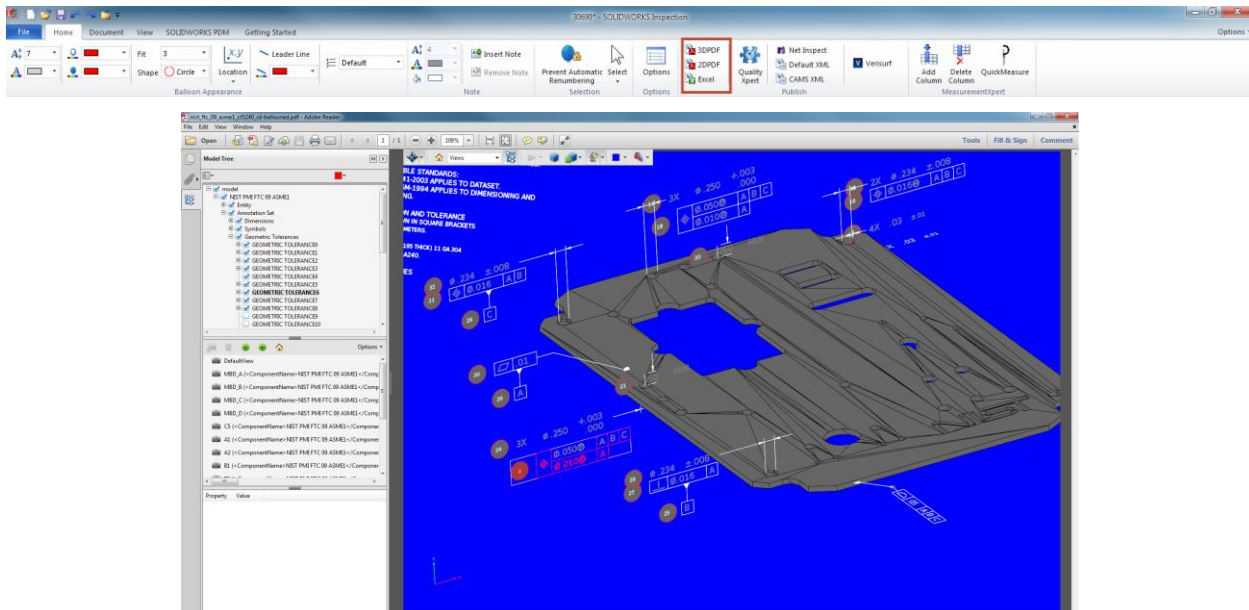
Benefits: By leveraging existing 3D CAD files users can streamline their quality control and documentation processes. The support of 3D data also enables companies to expand their drawing-less manufacturing strategy all the way to the quality department. Finally, this also eliminate the need to extract information from image type documents like PDF or TIFF files which could be significantly more time consuming.

2D FILES SUPPORT

Users can now open and import 2D CAD files from other CAD vendors directly within their inspection project. The following formats are supported with the 2018 release of SOLIDWORKS Inspection:

- AutoCAD® DWG files
- PTC® Creo Drawing files
- CATIA® V5 CATDrawing files
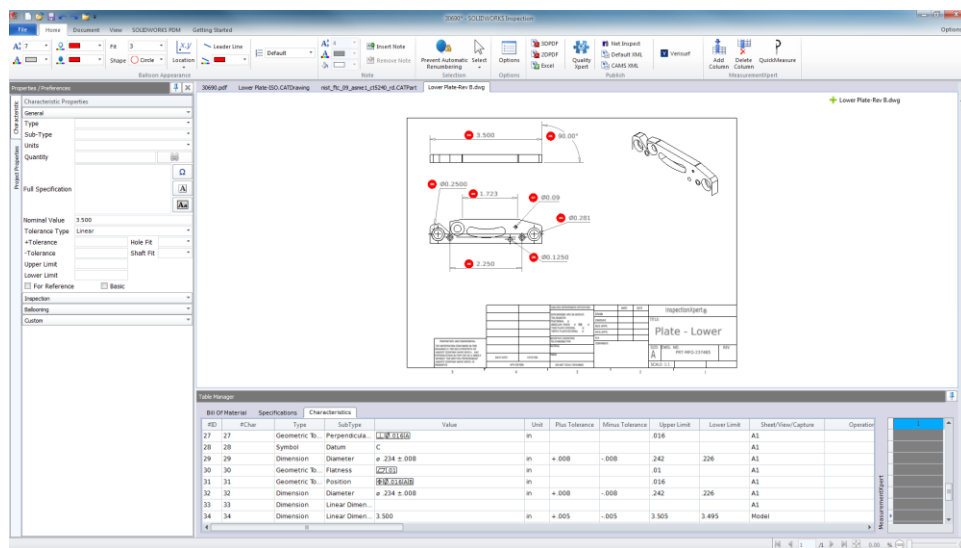
Similar to working with 3D files, users can use Smart Extract feature to extract one or multiple characteristics based on predefined settings.

Once the inspection project is completed, users can generate a Microsoft ® Excel file or a 2D PDF.

Benefits: Leveraging 2D CAD file speed up the process and enables users to create their First Article Inspection Reports (FAIRs) in no time while completely eliminating errors.

IMPROVED OPTICAL CHARACTER RECOGNITION (OCR) ENGINE

SOLIDWORKS Inspection 2018 features an enhanced OCR engine. Documents and fonts that were difficult to recognize are now extracted properly and with less input from the user.
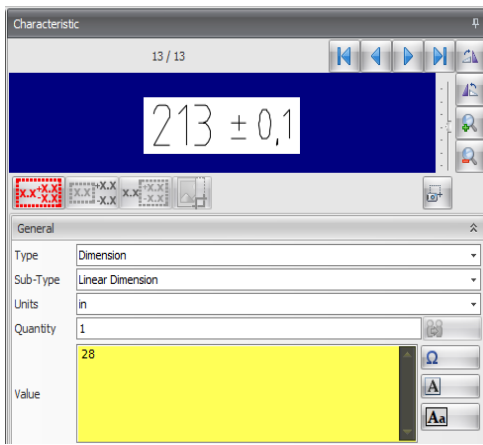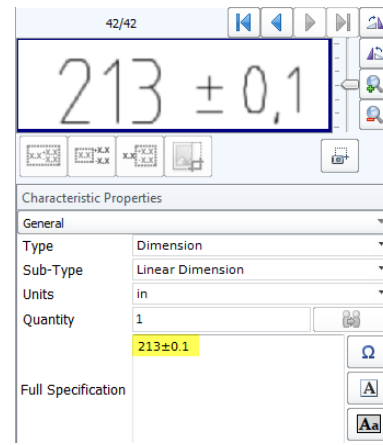
SOLIDWORKS Inspection 2017   SOLIDWORKS Inspection 2018

Benefits: Improved OCR engine reduces the time spent correcting extraction errors and limit the need to create custom OCR dictionaries.

SMART EXTRACT

Extracting information from a PDF file using the Optical Character Recognition (OCR) tool can be time consuming if the drawing contains dozens or hundreds of characteristics.

The new Smart Extract tool allows users to extract multiple characteristics at once if the PDF document used has a searchable text layer and support Smart Extract.

After selecting Smart Extract from the CommandManager or the Right Mouse Click menu, users can either decide to extract a single characteristic or multiple characteristics.

#ID	#Char	Type	SubType	Value
1	1	Geometric To...	Cylindricity	⌀ .025 A

This is particularly useful to extract complete GD&T without having to manually recognize each element.

Similarly, multiple characteristics can be extracted at once using pre-defined settings. The quality of the extraction depends on the software that was used to create the 2D drawing.
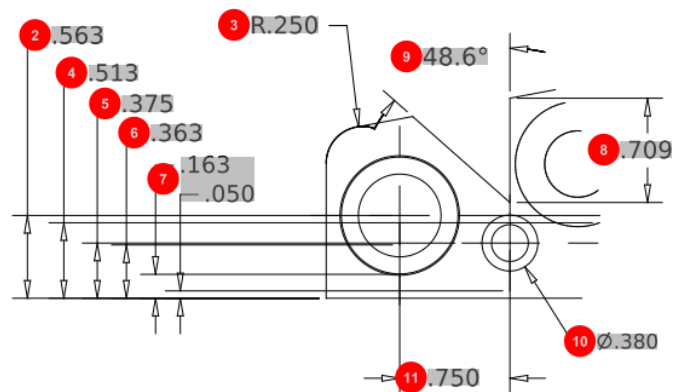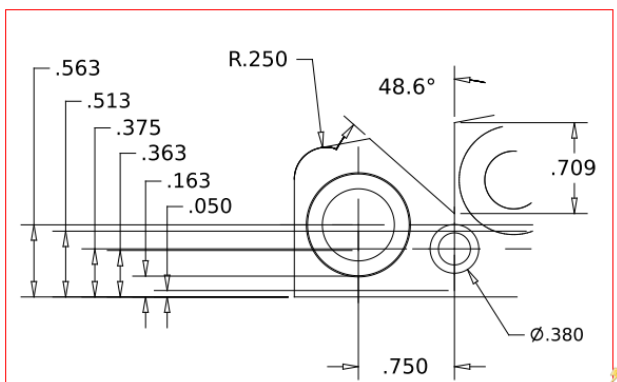
Bill Of Material		Specifications		Characteristics				
#ID	#Char	Type	SubType	Value	Unit	Plus Tolerance	Minus Tolerance
1	1	Geometric To...	Cylindricity	⌀ .025	in		
2	2	Dimension	Linear Dimen...	.563	in	+.005	-.005
3	3	Dimension	Radius	R.250	in	+.005	-.005
4	4	Dimension	Linear Dimen...	.513	in	+.005	-.005
5	5	Dimension	Linear Dimen...	.375	in	+.005	-.005
6	6	Dimension	Linear Dimen...	.363	in	+.005	-.005
7	7	Dimension	Linear Dimen...	.163 / .050	in		
8	8	Dimension	Linear Dimen...	.709	in	+.005	-.005
9	9	Dimension	Angle Dimen...	48.6°	deg	+1°	-1°
10	10	Dimension	Diameter	Ø.380	in	+.005	-.005

Benefits: Smart Extract dramatically speeds up the ballooning of drawings and creation of inspection reports.

EXTRACTIONXPERT(™)

The new ExtractionXpert(™) ![ExtractionXpert] tool inside SOLIDWORKS Inspection 2018 help users improve their Optical Character Recognition (OCR) readings by automatically trying different OCR settings such as Sharpen, Dilate, Edge Enhancement, etc. The ExtractionXpert tool goes over 200 combinations to recommend the best possible settings based on the values entered.

ExtractionXpert also accounts for scaling and therefore eliminates the need to manually rescale PDFs or TIFFs drawings. As a consequence, the "**Options > Project Options > Drawing**" tab has been removed in SOLIDWORKS Inspection 2018.
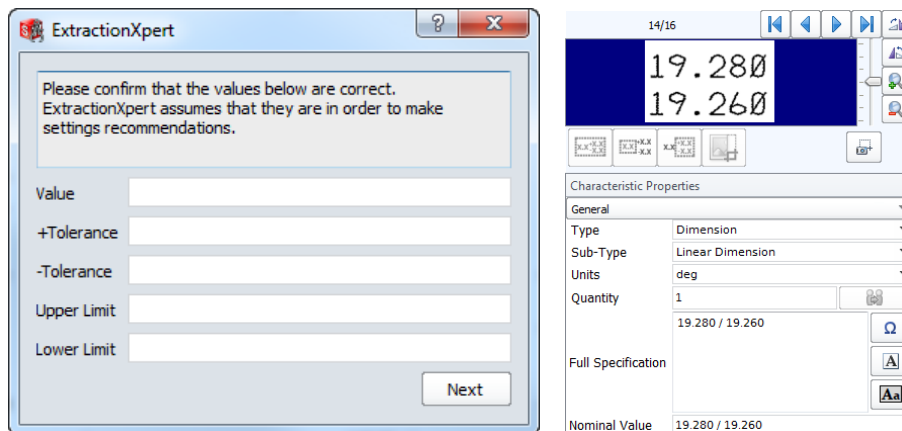
Benefits: ExtractionXpert helps users save time by finding the best possible settings for the OCR engine in order to extract information from PDF or TIFF documents with higher accuracy.

SAMPLING

Acceptance Quality Limit (AQL) is a method of sampling. It is adopted by many companies as the standard tool for choosing sample size for product inspection. Sampling and AQL can now be defined in the SOLIDWORKS Inspection Standalone application for the entire project or for each individual characteristic.

Benefits: SOLIDWORKS Inspection users now have an effective approach to plan their quality control processes without having to rely on other tools.
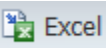
FULL SPECIFICATION

When extracting characteristics from a 2D or 3D file, the full specification is calculated in the characteristic window. In some companies, users are required to have the full specification (with tolerances, degree or diameter symbol, etc…) displayed in the inspection report.

Benefits: The new Full Specification field eliminates the need for some users to use complex Microsoft ® Excel formulas to recreate it using a combination of other fields.

PUBLISH REPORTS WITH DOCUMENT SNAPSHOTS

SOLIDWORKS Inspection Standalone 2018 adds new options and features to the Export to Excel functionality. Captures, views or drawing sheets can now be included directly within the Microsoft ® Excel inspection report.

Users can easily select with view or sheet to include and they can also manually create additional snapshots to be included. Finally, the snapshots can be re-ordered before being exported to Excel.

This new feature allows users to have all the information they need for quality control within a single document.
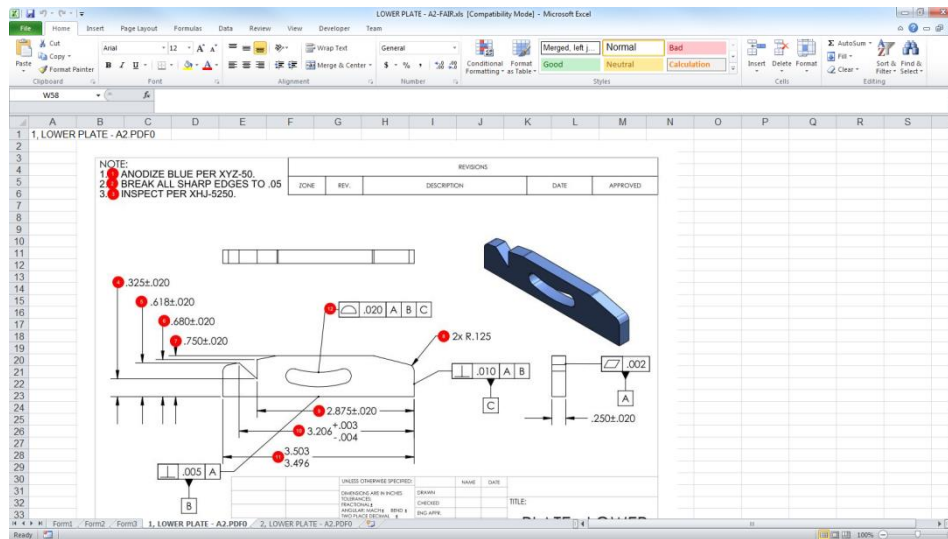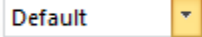
Benefits: This new option enable users to consolidate their ballooned drawings and inspection reports into a single document that can then be easily shared with inspectors in the quality department or suppliers.

BALLOON SEQUENCE

Balloon sequences can be created in the options and assigned to project's characteristics. New sequences can be added or removed using the [+] and [−] buttons and **Sequence** and **Starting Value** can be customized.

Characteristics ballooned by the user will be numbered based on the sequence selected in the CommandManager or in the characteristic window.

If the user select "Sequence 1", all new characteristics added will be numbered 100, 101, 102, … If a particular sequence already has characteristics, any new characteristic will be added to the end of the list (103 in this case).

Sequences can be modified in the options and balloon numbers will update accordingly.

Balloon sequencing, balloon increment and customizable balloon numbering can only be used independantly of one another in a project.

Benefits: Using the new Balloon Sequence or Customizable Balloon Numbering users can precisely balloon their documents without restrictions. With more flexibility, they can create exactly the report they are looking for.

CUSTOMIZABLE BALLOON NUMBERING

Using this new feature, users can now directly renumber characteristics in the Bill of Characteristic and introduce gaps if needed.

By double clicking on the Characteristic Number field, users can renumber it. All characteristics below it will be renumbered accordingly.

Table Manager

#ID	#Char	Type	SubType	Value	Unit	Plus Tolerance	Minus Toler
1	1	Note	Note	BREAK ALL SHARP CORNERS TO .05 INCHES.			
2	2	Note	Note	INTERPRET PER ASME Y14.5-2003			
3	3	Note	Note	HEAT TREAT PER GWS 1-08.			
4	4	Dimension	Linear Dimen...	88.9	in	+0.05	-0.10
5	5	Dimension	Linear Dimen...	6.4 / 6.3	in		
6	6	Dimension	Linear Dimen...	69.85	in	+0.5	-0.5
7	7	Dimension	Linear Dimen...	25.4	in	+0.5	-0.5
8	8	Dimension	Linear Dimen...	15.88	in	+0.5	-0.5
9	9	Dimension	Linear Dimen...	7.47	in	+0.5	-0.5

Table Manager

#ID	#Char	Type	SubType	Value	Unit	Plus Tolerance	Minus Toler
25	25	Note	Note	BREAK ALL SHARP CORNERS TO .05 INCHES.			
26	26	Note	Note	INTERPRET PER ASME Y14.5-2003			
27	27	Note	Note	HEAT TREAT PER GWS 1-08.			
28	28	Dimension	Linear Dimen...	88.9	in	+0.05	-0.10
29	29	Dimension	Linear Dimen...	6.4 / 6.3	in		
30	30	Dimension	Linear Dimen...	69.85	in	+0.5	-0.5
31	31	Dimension	Linear Dimen...	25.4	in	+0.5	-0.5
32	32	Dimension	Linear Dimen...	15.88	in	+0.5	-0.5
33	33	Dimension	Linear Dimen...	7.47	in	+0.5	-0.5

15	15
16	16
17	17

15	15
100	100
101	101
102	102
103	103

Balloon sequencing, balloon increment and customizable balloon numbering can only be used independantly of one another in a project.

Benefits: Using the new Balloon Sequence or Customizable Balloon Numbering users can precisely balloon their documents without restrictions. With more flexibility, they can create exactly the report they are looking for.
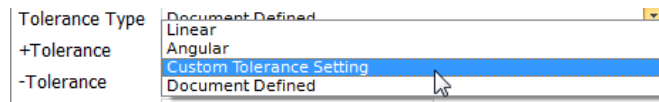
CUSTOMIZABLE TOLERANCE TYPE

Default tolerances can be customized and used if an explicit tolerance is not specified in the 2D or 3D file for one or multiple characteristics. Tolerance tables can be based on precision or range.

In addition to the Linear and Angular default tolerances, users can now add new tolerances. To start, users have to Click the ✛ and then they can double click on the tab to rename it.

Linear   Angular   Custom Tolerance Setting   +

	Precision	+Tolerance	-Tolerance
1	1	+0.5	-0.5
2	2	+0.05	-0.05

They can add and remove lines in the tolerance table using the [+] and [−]. They can also switch between "**by precision**" and "**by range**" using the radio buttons.

Once the tolerance table is completed, it can be easily selected in the characteristic window.

Benefits: Multipage drawings often have different tolerances defined on the title block of each page. The new customizable tolerance type feature enables users to quickly specific for one or multiple characteristics which tolerances should be applied.

CHARACTERISTICS IDENTIFICATION WITH BALLOON APPEARANCE PRESETS

New "**Characteristic Identification**" feature found within the **Ballooning Project Options** can be used to predefine a variety of balloon appearance settings that will be automatically applied based on either a customizable **Identification Criteria** or a selected **Classification**. This can be applied to Key Characteristics and therefore replace the Key Characteristics option (Identified and Placement) found in previous versions.
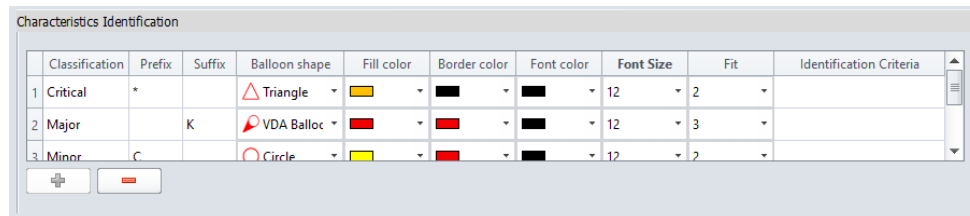
Benefits: Users can quickly apply unique balloon appearances to special characteristics for easy identification when performing an inspection.
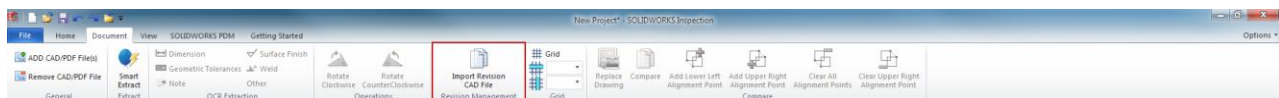
UNDO / REDO

Undo and Redo buttons are now available.

Benefits: Users can easily cancel their last action or redo it if necessary. This saves time and improves the user experience.

REVISION MANAGEMENT FOR 3D CAD FILES

The new revision management tool allows users to automatically analyze changes between two CAD files and provides a summary of all annotations that have been removed, modified, or added.

Users can choose to have SOLIDWORKS Inspection automatically process all of the changes between revisions, or to highlight all changed items in the project, allowing the user to review each item individually.

Using the manual revising processing mode provides a summary of the changes as well as instruction to make the changes. On the other hand, the automatic mode displays a summary with options and user preferences for automatic processing.

#ID	#Char	Type	SubType	Value	Unit	Plus Tolerance	Minus Toler
16	16	Symbol	Datum	C			
17	17	Symbol	Datum	A			
18	18	Dimension	Linear Dimen...	1.723	in	+.005	-.005
19	19	Geometric To...	Profile of a S...	⌒ .020 A B C	in		

Bill Of Material | Specifications | Characteristics

Modified and removed characteristics are highlighted orange and red in the table manager while removed, modified and new annotations are highlighted in red, orange and green in the graphic area.
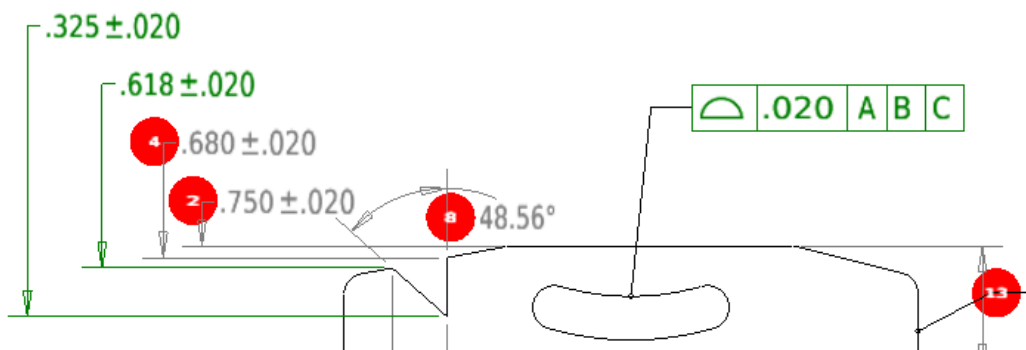
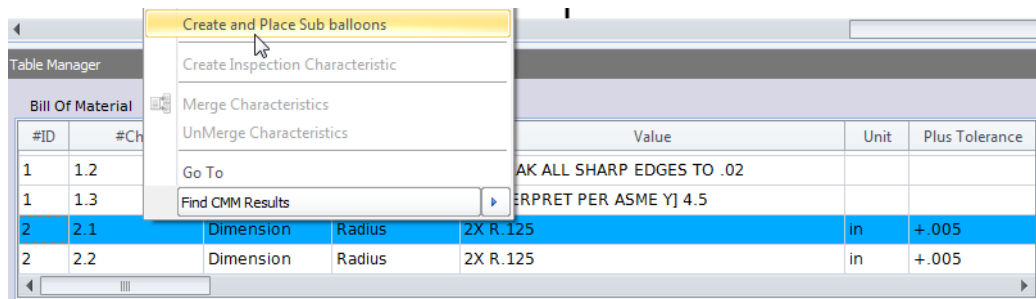**Benefits**: Users can manage revisions with ease regardless if they are working on a 2D document or a 3D CAD file.
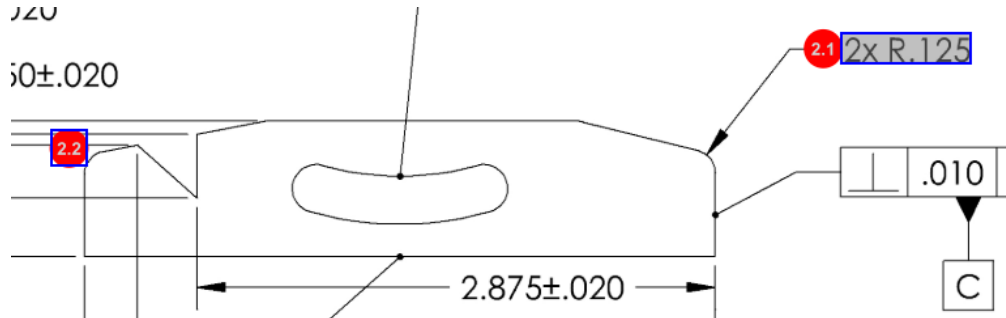

CREATE AND PLACE SUB BALLOONS

Some characteristics can refer to multiple features of a part. For example, this radius refers to 2 fillets.

After creating a line for each instance , users can now very quickly add multiple balloons on the drawing. Using the contextual menu, users need to first select "**Create and Place Sub Balloons**".

Table Manager							
Bill Of Material							
#ID	#Ch			Value	Unit	Plus Tolerance	
1	1.2			AK ALL SHARP EDGES TO .02			
1	1.3			ERPRET PER ASME Y] 4.5			
2	2.1	Dimension	Radius	2X R.125	in	+.005	
2	2.2	Dimension	Radius	2X R.125	in	+.005	

Then, they can click anywhere in the document to add the additional balloons. In this case, only balloon #2.2 with balloon #2 being renumbered #2.1

Benefits: The new "Create and Place Sub Balloon" feature streamlines the creation of sub-balloons saving and trouble to SOLIDWORKS Inspection users

EXPLODE NOTE

Drawing's notes can sometimes contain multiple characteristics to inspect. If that's the case, users need to be able to explode a note into multiple instances.
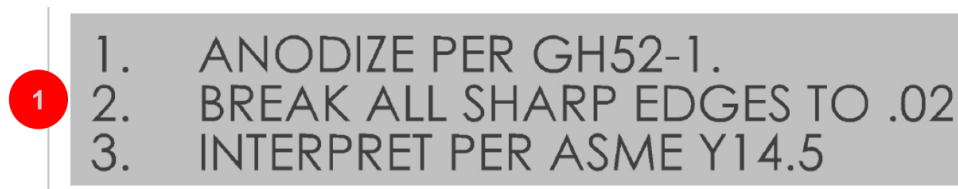
In the Table Manager, using the contextual menu, select "**Explode Note**".

Then, select each characteristic and create a new instance or use "Auto-Explode".

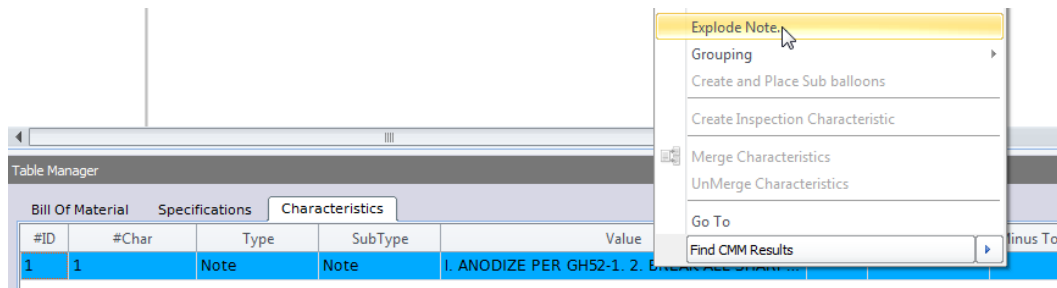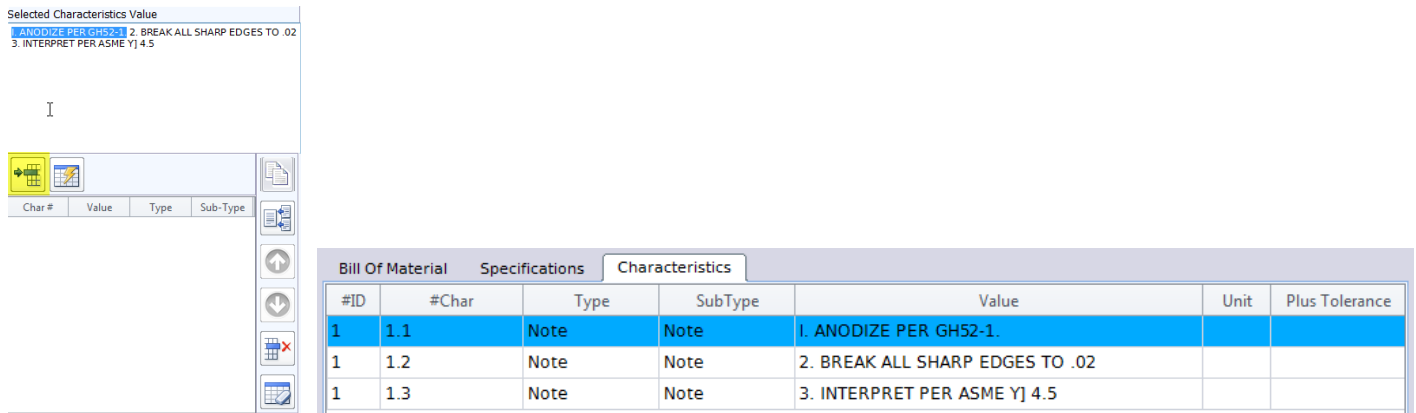| #ID	#Char	Type	SubType	Value	Unit	Plus Tolerance
1	1.1	Note	Note	I. ANODIZE PER GH52-1.		
1	1.2	Note	Note	2. BREAK ALL SHARP EDGES TO .02		
1	1.3	Note	Note	3. INTERPRET PER ASME Y] 4.5		

Users can also add, remove, copy or combine rows if needed.

Benefits: Users can easily explode notes into separate instances for their inspection reports.

MERGE CHARACTERISTICS

In SOLIDWORKS Inspection 2018, multiple characteristics can be merged to create a single one.

In the Table Manager, select the characteristics to merge and right mouse click to display the contextual menu. Then, select "**Merge Characteristics**"
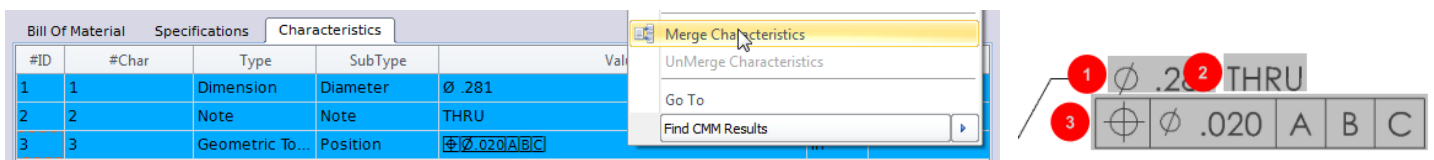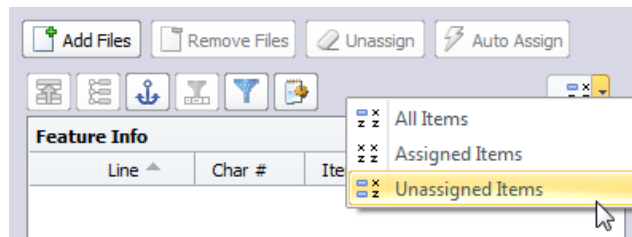

The characteristics are then combined into a single one.

#ID	#Char	Type	SubType	Value
1	1	Dimension	Diameter	Ø .281 THRU ⊕ Ø.020 A B C

Benefits: Users have more controls over the look and feel of their inspection report.

CMM DATA MANAGEMENT IMPORT

New filters are available to help quickly identify items that are assigned or unassigned.


Benefits: Users can import and assign their CMM result files faster with new filters to quickly highlight unassigned items.

IGNORE BASIC DIMENSION MEASUREMENTS

Basic dimensions are often included in inspection reports. They can be used, for example, to determine if a hole is an acceptable position in combination with other characteristics.

In SOLIDWORKS Inspection 2018 Standalone, users can decide to ignore them from measurements input so they do not get flagged (as either in or out of tolerance)
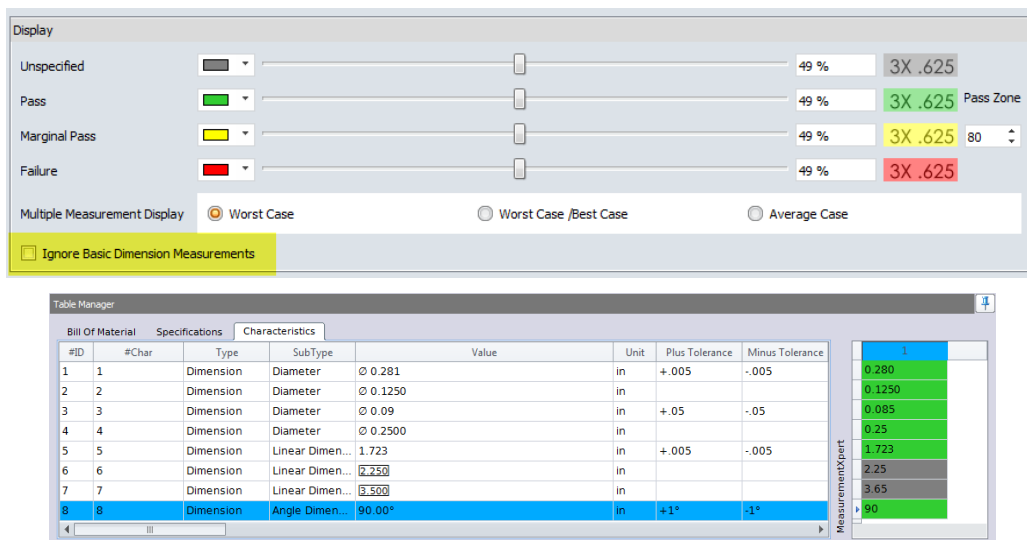

Benefits: Users can now ignore basic dimension measurements.

PDM INTEGRATION

SOLIDWORKS PDM integration is now part of SOLIDWORKS Inspection Standalone 2018. Users can access SOLIDWORKS PDM features – such as Check Out, Check In, Search, Version, etc. – using the PDM tab in the CommandManager.

This new integration helps quality engineers manage and centralize the storage of their inspection projects as well as all the files and reports related to those. Additionally, version control helps prevent data loss and the search feature dramatically reduces the time spent searching for inspection projects.
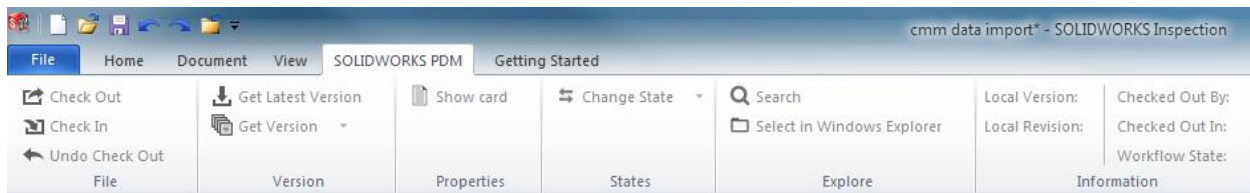
Benefits: Thanks to the SOLIDWORKS PDM integration, users can manage their overall product development with ease and include quality management as an integral part of the process.
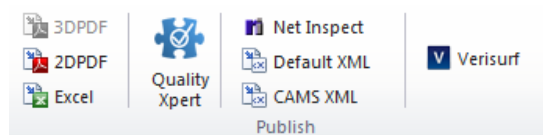
ADDITIONAL NET-INSPECT OPTIONS

Net-Inspect is a web based quality management system. It is used by many large companies to manage their inspection processes both internally and with suppliers worldwide.
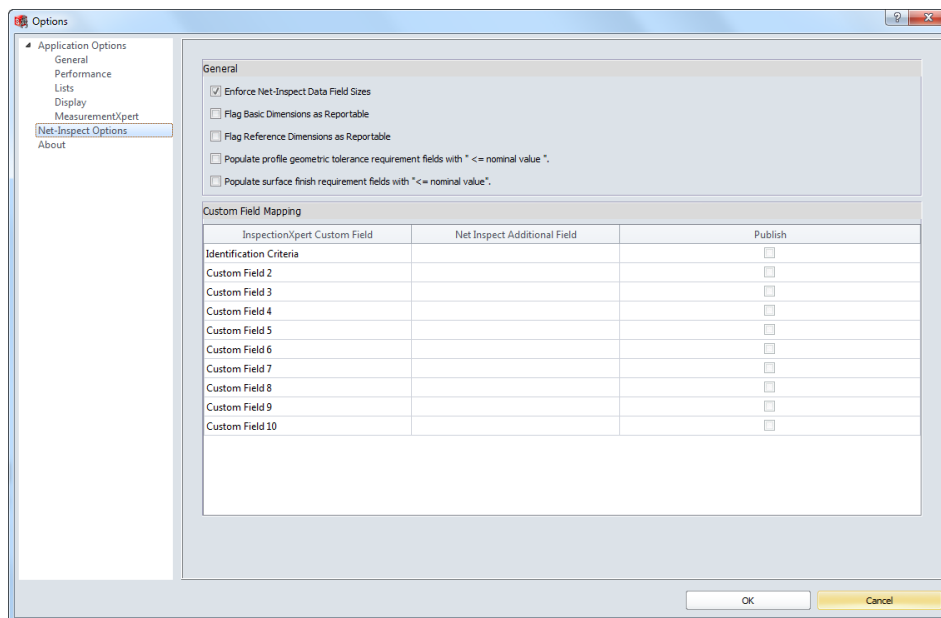
Benefits: Users can now define more export options and parameters when exporting their inspection projects to Net-Inspect.

EXPORT TO CAMS XML AND VERISURF

Additional export options have been added to SOLIDWORKS Inspection 2018. Users can now export their inspection projects to CAMS XML or Verisurf.

Benefits: Additional export options allow users to easily share their SOLIDWORKS Inspection projects with other quality management tools.

EXPORT TO QUALITYXPERT IMPROVEMENTS

QualityXpert is a shop floor manufacturing intelligence solution that enables small and medium-sized manufacturers to manage quality and gain actionable insights in real-time from their shop floor.
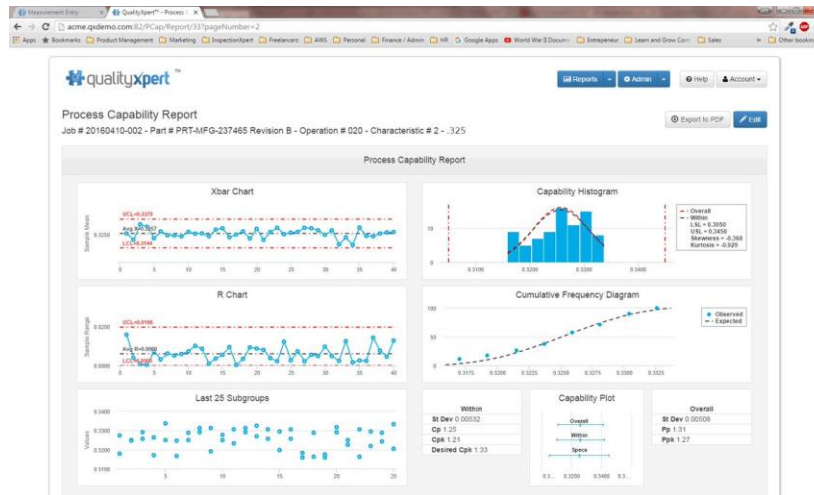
Benefits: Users may now publish their inspection plans to the web for inspection on the shop floor via mobile device.