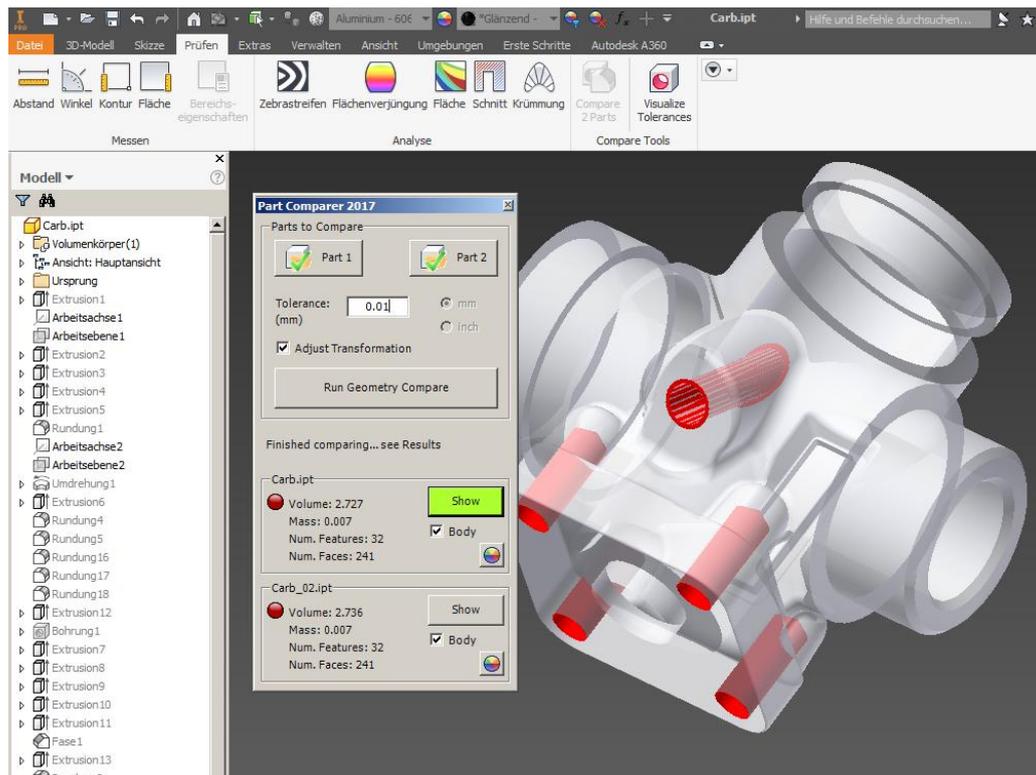


Part-Compare + Tolerance Visualize AddIn Manual (Inventor 2016, 2017, 2018)



This manual describes the installation and the usage of the Part-Compare and Tolerance Visualize AddIns for Inventor 2016, 2017 und 2018. Please read this manual carefully before using the Application.

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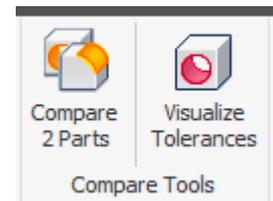
1. Installation

The installation of the Part-Compare AddIns for inventor 2015 in 2016 or 2017 is simple: Quit Inventor and copy all the files from the zip file to (xxxx is your Inventor version):

C:\ProgramData\Autodesk\Inventor xxx\Addins

2. Start Buttons

If no files are opened in Inventor, you will find the buttons for the Part-Compare and Visualize Tolerances AddIn in the tab 'First Steps' in panel 'Compare Tools' available. However, are in Inventor files already open you can find the button for the Part-Compare and Visualize Tolerances AddIn in the tab 'Test'.



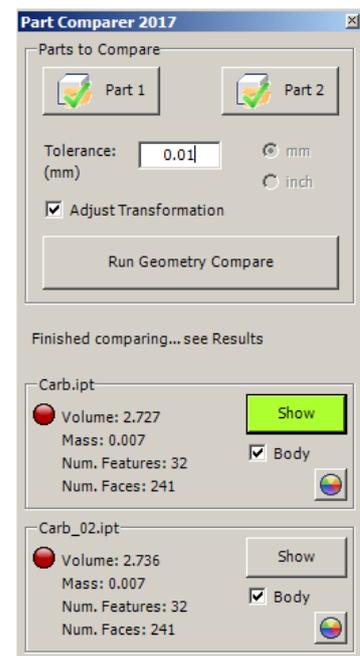
3. Define part files to compare

Is a Part file opened in Inventor it will be loaded at the start of Part-Compare AddIn immediately as the first to compare file.

Are two Part files open these are loaded by default at the start of Part-Compare AddIns. Alternatively, at any time another Part file can be opened via the file dialogue by on one of the buttons 'Part 1' or 'Part 2' a file. The names of the open files are displayed in the two summary fields.

A great advantage of the Part-Compare AddIn is that not only solids can be compared. The Application easily handles even surface models and combined Surface and Solid body models. The Application processes all CAD formats that can be loaded in Inventor.

The Part-Compare AddIn automatically detects whether it is a metric or inch model. Accordingly, the tolerance for evaluation in mm or inch is indicated. Are the models/geometries in different measuring systems, the user decides whether the comparison should be made in mm or inch.



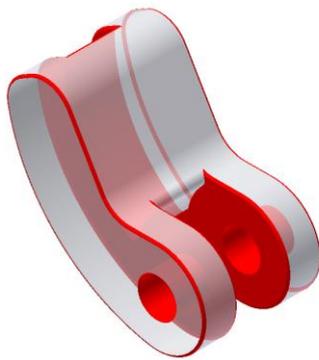
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4. Tolerance

A unique feature of the Part-Compare AddIn is to define a tolerance for the comparison of 2 components. Only the differences that lie over the desired tolerance value will be displayed. Thus it can be seen to what extent certain areas differ from 2 components by calculating a different tolerance is repeated. Example:

Tolerance 0.01mm

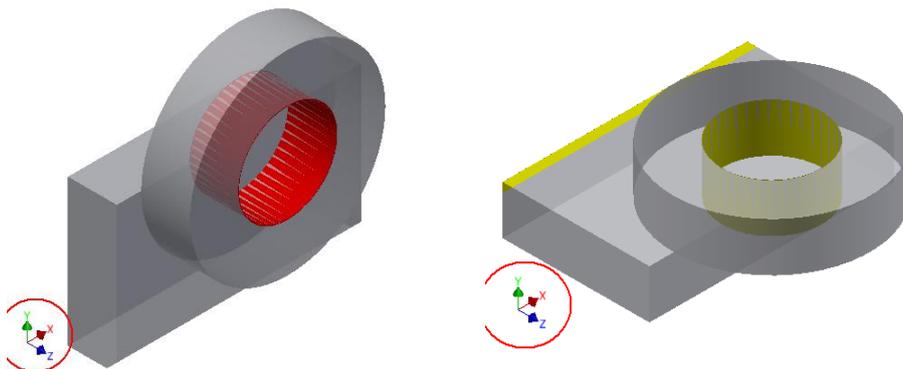


Tolerance 0.05mm



5. Parts with a different orientation

Another unique feature for the comparison of geometries is the ability to reliably compare two components with different orientation in space together. To do this simply 'Adjust Transformation' must be switched on (default). The Part-Compare AddIn overlays the two models to a best fit to compare both model. The output of the calculation results are in the appropriate matrix, so that the differences in the two models are displayed correctly. Example:



Restriction: by using too simple models - such as cylinders or blocks - this function can supply an incorrect result back.

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6. Checking Part history

The Part-Compare AddIn compares the two components in terms of volume, mass, number of feature and number of faces. Differing values are highlighted with a red dot for easy recognition.

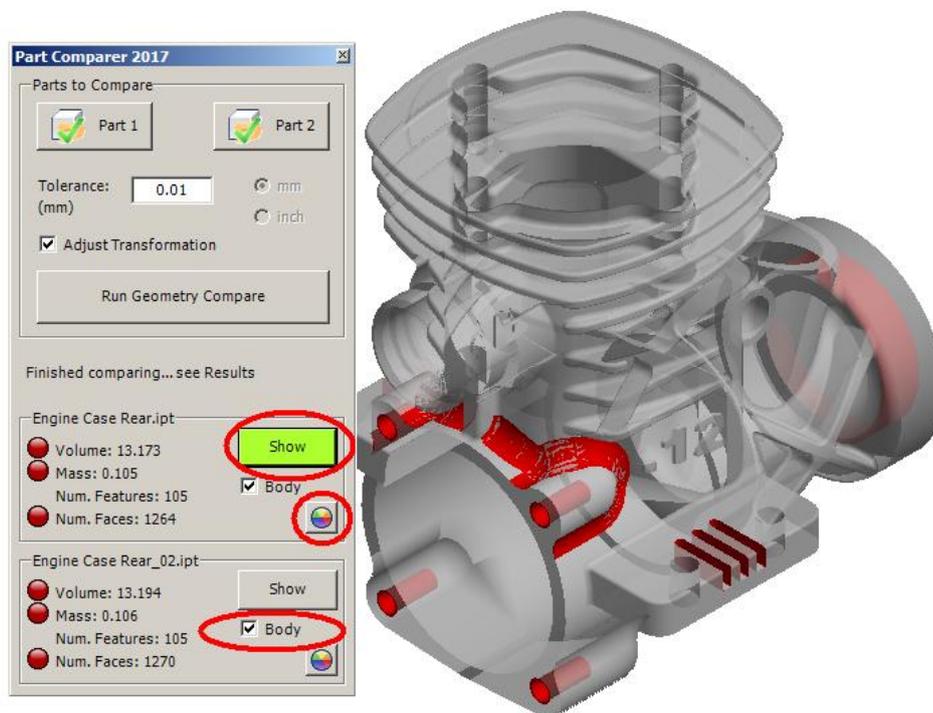
7. Checking Part geometry

Detects the Part-Compare AddIn that due to the input tolerance, the two components are identical, no result can be displayed. If, however, found that the two components differ, the respective differences can be displayed by clicking the button 'Show'. The coloring of the base geometry changes in a semi-transparent gray. The differences are highlighted in a selected color. With the check box 'Body' the displaying of the body can be switched on or off. The color to display the differences can be set via the color selector.

You can switch back and forth between any of the two models by 'Show', 'Body' or the color selector is clicked.

How is this colored highlighting to understand?

The Part-Compare AddIn raises in each component include all surfaces and areas that do not exist due to the specified tolerance in the other component.



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8. Visualizing Part tolerances (new)

Inventor provides the ability for the device modeling to assign tolerances in parameters. Often it is unclear what the precise impact several tolerance parameters on the behavior of the entire component geometry can have. Depending on the type of dimensioning tolerances, it can generate undesirable side effects, so that the result between the minimum and maximum dimension does not fit with the original idea of the designer or may adversely affect the function of the component in an unexpected way.

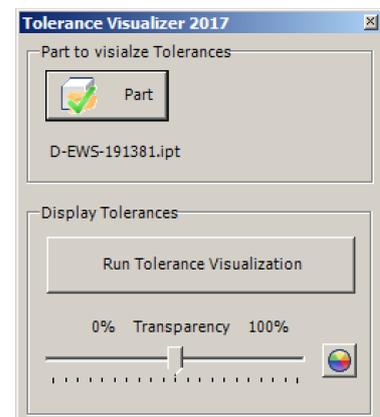
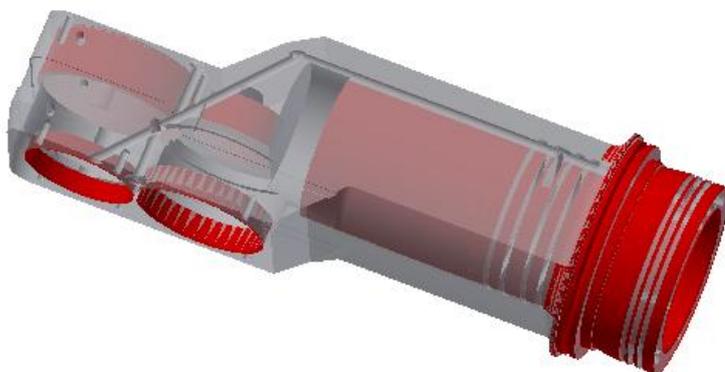
Likewise, with the help of this useful tool Designers can check Part components quickly and easily as to which areas include tolerances or which areas of component tolerances are to what extent affected.

The Visualize Tolerance AddIn Start button is also located on the Panel 'Compare Tools' (see Pt. 2). If a Part file is opened in Inventor it will be loaded at the start of the Visualize Tolerance AddIn immediately as the file to be checked. You may also select another Part file at any time by clicking on 'Part'.

The button 'Run Tolerance Visualization' calculates a difference model between minimum and maximum size of all to the component assigned tolerances and displays the result graphically. The Visualize Tolerance AddIn also takes care of the tolerances of derived components in the Part - even across several levels.

With the checkbox 'Body' switch the representation of the body is on or off. The color to display the differences can be set via the color selector.

Within a few seconds the Designer is served by a detailed overview of how and where component tolerances have an influence on the model and whether the granting of component tolerances on the 3D model is complete.



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