MDTools® 780 SP 02

Applicability: MDTools 780 with or without SP01, Hotfix 04, Hotfix 05 and Hotfix 06

Issues Resolved:

1. Unable to select another cavity as Origin for reference in Move Cavity command.

2. In Feature Mode when selecting all Bolt Holes of a Footprint from MDTools ribbon for suppressing, only one Bolt Hole suppressed in the Footprint

3. In Feature Mode When Suppressing a Bolt Hole of Footprint, attached engraving with Drill Holes got deleted.

4. Update Drawing command gives error, When MDTools Machining Chart modified manually.

5. Measure Distance Command displaying incorrect value for intersecting cavities.

6. When no read permission is granted for C:\ProgramData Settings path, MDTools is giving exception while creating Machining Chart

7. Machining Callout command displaying 'Cavity Data' error, when clicking more than once on 'Create' option in dialog.

8. When Option "Use Default Object Style" in Layers is unselected, wrong layer assigned for Grouped Bolt Holes.

9. Automatic Update Drawing causes Inventor to become unstable when Engraved Text Chart columns are manually Modified (Added/Deleted).

10. Convert Block command throwing errors in some cases when Engravings are attached to cavities.

11. Automatic Update Drawing Prompt not displaying when modifying the location of Engraved text by entering the value in X/Y option.

12. When there is only Isometric View in the drawing, Machining Chart is giving exception error.

13. With Inventor 2021, unable to select Origin or axis on selecting 'Change Origin' from 'Move Cavity' dialog.

14. Unsuppressing Cavity from Cavity browser is not working for older MDTools version created block.

15. Replace command removing Assembly outline of Footprints.

16. Compare Manifolds command reporting imcorect value of Drill Depth for Sun Cavities.

17. When a drill hole of a Footprint is inclined, MDTools is removing all orings attached to that Footprint.

18. Inserting a Footprint with Mandatory orings display error message.

19. MDTools shows errors in the Inventor model tree when first suppressing and then deleting drills from footprint.

20. When editing the net info of a "sun cavity" without STEP12 in Edit Cavity command, changes the LS depth of Sun Cavity.

21. Make Footprint returning unhandled exception error and not working in some specific cases.

22. When the "Move" command used on a Slip-In cavity, MDTools throws exception in some specific cases

23. Automatic Machining Drawing creation causes instability if the Drawing template contains additional sheets

24. Inserting O-ring Groove as Counterbore throwing exception with certain libraries

25. Stretch & Incline a drill in Footprint will not replace the entire footprint

26. Footprint Envelope gets deleted when replacing a Footprint by another Footprint on exporting to mbxml and importing the mbxml

27. Inserting O-ring Groove gives DBNull error for some specific Libraries.

28. Un-suppressing the Bolt Hole of a rotated Flange, Flange gets rotated to 0 Degree and also deletes the O-ring of another Flange.

29. After recreating the MDTools views, Auto dimension command not working.

30. Unhandled exception displays when opening a Part file in MDTools checked out from VAULT Server as Read Only.

31. When MDTools Add-in loaded with Tube & Pipe Addin, Inventor 2021 crashes while defining the Route inside Assembly.

32. When MDTools Add-in loaded with Frame Generator Addin, Inventor crashes while in Insert Frame command for assembly.

33. When MDTools Add-in loaded, Derive assembly' from assembly throws exception.

34. When orifice is connected to other Cavities, Parent cavity of Orifice not moving with Orifice.

35. After Un-suppressing a suppressed Bolt Hole in Rotated Footprints, location of BH is displayed incorrectly

Date: Sep 2, 2021

* MDTools® 780 SP 02 includes all earlier Hotfixes and Service Pack

PTO

MDTools® 780 Hotfix 06

Applicability: MDTools 780 with SP1 and Hotfix 05 installed

Issues Resolved:

1. Assemble Component command did not display Library subfolders that did not directly contain ".ipt" component files.

2. The "&" symbol was not allowed as a valid character for Component ID.

3. Mirror Footprint command was not creating Bolt Holes with Counterbore on the correct face.

Date: Jul 12, 2021

MDTools® 780 Hotfix 05

Applicability: MDTools 780 with SP1 installed

Issue Resolved:

1. When Bolt Hole of a Footprint is the parent cavity and a plain drill, then Stretch Drill and Edit cavity throws error.

2. Inclining the parent cavity of Footprint contaning 3 Step Bolt Holes throws error

3. In Feature Design Mode, when editing a 3-Step Bolt Hole in a Footprint, the Edit command displays error

4. In Stretch Drill command, when editing the depth for Ungrouped Single-Step Bolt Hole in a Footprint, it displays error

5. When opening the Insert Construction Port command there is random slowness.

Date: May 2, 2021

MDTools® 780 Hotfix 04

Applicability: MDTools 780 with SP1 installed

Issue Resolved:

1. Connect Cavity command relocates the connecting cavity although it is already aligned to the Footprint child cavity.

Date: Feb 16, 2021

MDTools® 780 SP1

Applicability: MDTools 780 with or without Hotfix 01, Hotfix 02, Hotfix 03

Issues Resolved:

1. For an inclined Expander Plug, the Assemble Plug command displays an error

2. HyDraw Schematic Interface: The '&' character in the engraving text does not display in the Cavity Information section.

3. Configure Machining Chart Options: The '&' character in the engraving text displays as long text for Machining ID, when you select the Machining ID same as Component ID option.

4. An incorrect warning message appears on dragging the center point of a cavity without selecting the connecting cavity port.

5. Update Drawing: The undercuts' Horizontal dimensions are not deleted when you update the drawing after deleting the Slip-In with the undercut from one face.

6. Metric Drawing: Show Machining ID and Show Component ID displays text with text height in Inch value.

7. Automatic Machining Drawing command: The selected sides for dimensioning does not match with the Auto Dimension command of 2D Drawing.

8. Machining Callout not displayed for a cavity on chamfered face

9. Balloon not displayed for a cavity on chamfered face

10. Part Environment: Design Check command prompts "Show Selected Net" on the top-left side of the Title bar.

11. Locate Cavity command: Unsorted order of Component ID and Machining ID.

12. Stretch Drill command: The Serial number column is not present in the Select Tool dialog box

13. For some specific templates, Automatic Machining Drawing displays an error

14. Converting block from the Feature mode to Virtual mode throws an error: face not found issue for Chamfered face cavity.

15. Wall Thickness Check displays incorrect results for blocks with cutouts.

16. Manufacturing and Tooling Check command reports Suppressed cavities as errors.

17. Delete command does not delete all cavities of a footprint if created using the Mirror Footprint command.

18. Replace command does not preserve the rotation angle when the footprint is replaced by selecting the Replace All similar Cavities option in Feature mode.

19. When Location precision is '0' in MDTools Option, Angular Connection shows a correct preview, but connection becomes offset once the command is completed.

20. Updating assembly throwing an error when a cavity with orifice plug face is changed or suppressed.

21. When using Vault Server for a particular assembly, Assemble Component throws an error.

22. In a few specific cases, Inventor freezes when updating dimensions of cavities using the Update Drawing command.

23. Drawing Layers: MDTools checks for layer availability even if the Use Default Object Style option is unchecked.

24. In a few specific cases, Convert from Virtual to Feature causes Inventor to freeze when rotated footprints are present in the Part model.

25. O-ring Groove inserted as CounterBore does not update the color of a parent cavity

26. Unable to create a circular pattern around a work axis.

27. When a flange is defined as a Construction Port, then the Assemble Plugs command does not assemble the CAD Model on the flange.

28. Configure Auto Dimenison: Dimensions do not get added when the Auto Dimension command is rerun for different Cavity Type selections.

Date: Jan 12, 2021