INTEGRATED PARTNER SOLUTIONS, INC.

MechWorks PDM R17

The **Complete Solution** for SolidWorks®, Autodesk Inventor® and Siemens Solid Edge® documents management.

MechWorks PDM R17 Getting Started

NOTE: This Getting Started Guide is not a substitute for training

MechWorks PDM R17

Getting Started

MechWorks PDM is developed and owned by ***MechWorks***.

This manual was developed by ***Integrated Partner Solutions, Inc.*** and certified by ***MechWorks***.

©2001-2017 Integrated Partner Solutions, Inc.
PO Box 277
Hammond, WI 54015 USA
All Rights Reserved

The information and software discussed in this document is subject to change without notice.
None of its contents may be reproduced or transmitted in any form or by any means without the express written permission of ***Integrated Partner Solutions, Inc.***

SolidWorks® is a registered trademark of SolidWorks Corporation.
Inventor® is a registered trademark of AutoDesk Corporation.
Solid Edge® is a registered trademark of Siemens Corporation.
Microsoft® is a registered trademark of Microsoft Corporation.

INTEGRATED PARTNER SOLUTIONS, INC.

Integrated Partner Solutions, Inc. is a North American distributor of MechWorks PDM and other applications produced by ***MechWorks***.

MechWorks s.r.l.
via Vallescura 8/2
40136 Bologna
Italy

www.mechworks.com

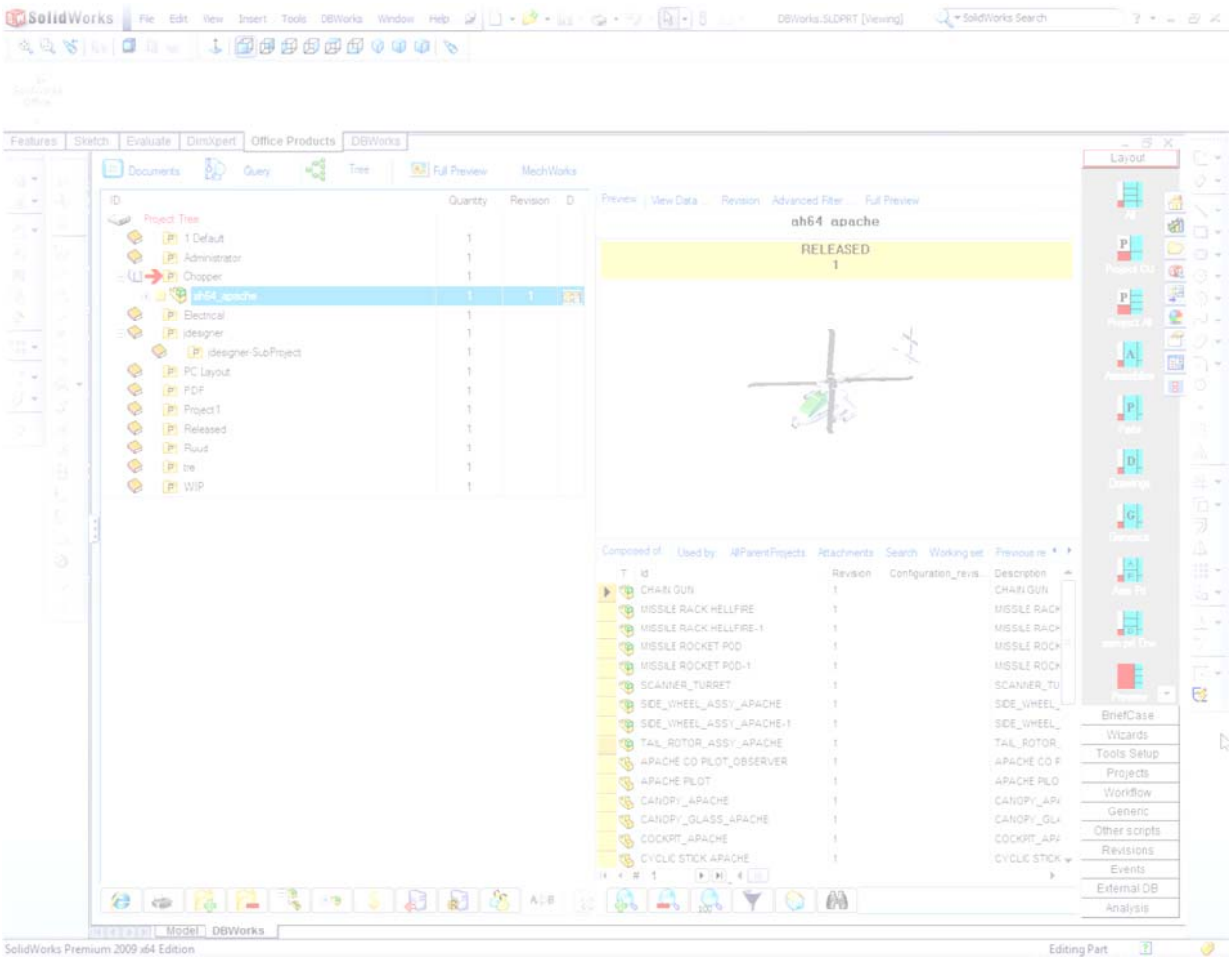
Integrated Partner Solutions, Inc.
PO Box 277
Hammond, WI 54015
USA

www.na-ips.com

TABLE of CONTENTS

Chapter 1

What is MechWorks PDM



What this Getting Started Manual offers

This manual is a brief overview of MechWorks PDM. It is not intended to be a substitute for MechWorks Administrator Training or Essentials Users Training. This manual has some very basic instructions for the installation, overview and basic running of MechWorks PDM.

MechWorks PDM Administrative training covers the Administrator Functions and setup of MechWorks PDM in-depth. The details of how to install MechWorks PDM, how to do basic Customization and what the Options are and how they work is something that is critical to a company's ability to get the most of MechWorks PDM.

MechWorks PDM User Essentials training covers the essential commands for day to day use of MechWorks PDM. This training class is designed to demonstrate how CAD users as well as Standalone users interact with MechWorks PDM on a daily basis.

For more information on either the MechWorks Administrator Training or Essential Users training, please contact your MechWorks PDM Reseller.

Screen shots within this manual are taken from a Windows 7/8 operating system.

The use of the term **CAD** refers to SolidWorks, Inventor or Solid Edge.

 Represents areas that pertain to DBWorks.

 Represents areas that pertain to DBInventor.

 Represents areas that pertain to DBSolidEdge.

 Represents areas that pertain to Standalone.

What is MechWorks PDM?

MechWorks PDM is the Complete solution for SolidWorks®, Autodesk Inventor® and Siemens Solid Edge® documents management. As an out-of-the-box solution, MechWorks PDM supports editing of checked out documents in the context of assemblies so that the user does not sacrifice CAD functionality. Scalable from an SQL Express database to a Client/ Server SQL Server, MechWorks PDM provides the solution for small CAD work groups to large enterprise installations.

MechWorks PDM provides an open API to allow users to customize MechWorks PDM to work in an already established environment, rather than having to customize your environment to fit with MechWorks PDM.

Information output in Excel or HTML formats, provide the users a standard way of sharing information across your company's applications. Used-by and Composed-of information is displayed by simply selecting a document's record. Additional documentation formats such as DOCX, DXF, DWG, BMP etc. can be linked to any Project or CAD document and managed automatically.

MechWorks PDM allows you to determine the amount of control over your documents that you wish to maintain. A Standalone version is also available for users that do not have CAD and need access to the database, documents and information related to those documents.

Project centric layout

- Explore all of the documents from the project tree
- View Sub-projects
- Define project visibility
- Expand and collapse any project tree node
- Consistent user interface for all actions
- Drag & Drop to move or copy documents between projects
- Drag files from Windows Explorer to add documents to projects

MechWorks PDM briefcase

Exchange documents with remote collaborating companies
MechWorks PDM can save into a single .zip file:

- A selected set of documents
- All the previous versions of the selected documents
- A subset of the database containing all the related information

Eventually, MechWorks PDM can retrieve the documents and database information from a single file and merge the content in your database and file system seamlessly.

Local checkout mode

- Define a separation between work-in-progress documents and released documents
- Ensure that viewers have access only to released documents, while designers work on separate data and documents

Geographically distributed MechWorks PDM sites

For subsidiaries linked by high speed connections, MechWorks PDM enables data sharing through a centralized SQL Server database and handles documents' ownership.

MechWorks PDM Web Client

- Query your database via the internet.
- Download compressed files, PDFs and DXF files.
- Preview your SolidWorks/Inventor/Solid Edge/AutoCAD documents in the viewers.

More

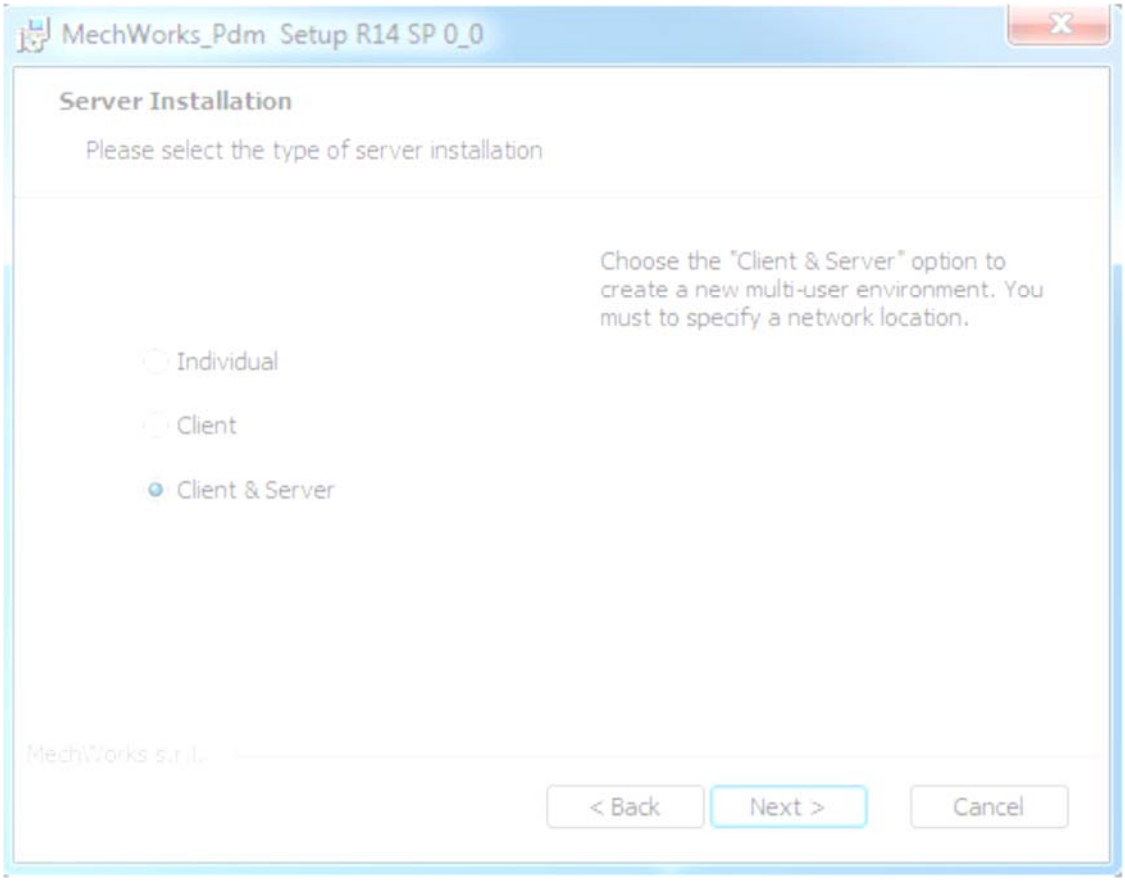
- Improved Word/Excel integration
- Bill of Materials for any given previous version
- Improved dynamic interface in popup menus
- Batch plotting available also in Standalone version
- Sort browser tables by double clicking the field header
- Change the layout dynamically using buttons to adapt to the documents you're viewing
- Database fields updated from custom properties
- Form designer tool to display your own forms and handle events via scripting
- Improved registration speed of assembly structure in the database
- Improved preview of generic documents
- New options
- Improved options management

The following lessons of this Getting Started manual just scratch the surface of what is offered in MechWorks PDM. For additional information regarding functionality, options, installation methods, etc., see the MechWorks PDM Help files. Additionally, MechWorks PDM Administration Manuals and Essentials User manuals are available. For more information please contact your MechWorks PDM Reseller.


Intentionally Blank,
as a separator.

Chapter 2

How to Install MechWorks PDM



Installation of MechWorks PDM

The installation of MechWorks PDM is similar to the installation of SolidWorks\Inventor\SolidEdge. You need to consider your current networking environment as well as the number of users that will need access to your MechWorks PDM information. MechWorks PDM has multiple methods available for installation. These lessons will outline the different methods available. MechWorks PDM supports two different databases*, SQL Express and Microsoft SQL Server 2005®/2008®/2012®/2014®. **Note: The method you choose for your installation should be planned out ahead of time.**

Pre-Installation considerations

There are a number of items to consider prior to installing MechWorks PDM; this manual will cover many of these, but not all of them. Some considerations are: Operating Systems, Database types, pre-requisite software, version of the CAD installed and MechWorks PDM Registration Codes. **Note: This manual does not directly cover the installation of Databases or Operating Systems. It is assumed that both have already been completed prior to installing MechWorks PDM.**

Operating Systems

MechWorks PDM is certified to run in the same Microsoft operating systems as the version of CAD installed. Please reference the following URL for updates regarding supported environments:

http://www.mechworks.com/Compatibility.asp

Supported Databases

MechWorks PDM supports the use of the following databases:

- SQL Server 2008R2 Express
- SQL Server 2008 R2
- SQL Server 2012 Express
- SQL Server 2012
- SQL Server 2014 Express
- SQL Server 2014
- SQL Server 2016 Express
- SQL Server 2016

*Microsoft Access is used with the MechWorks PDM Briefcase functionality.
**Microsoft Windows Server 2008 is supported only in server mode (as the repository for MECHWORKS PDM_SERVER shared folder and MechWorks PDM Standalone). DBWServer is also supported on such platform.

Pre-requisite software

MechWorks PDM (Integrated and Standalone) needs to have access to Microsoft® databases, namely SQL Server 2008®/2012®/2014®/2016®. This implies the presence on the computer of the database access components from Microsoft.

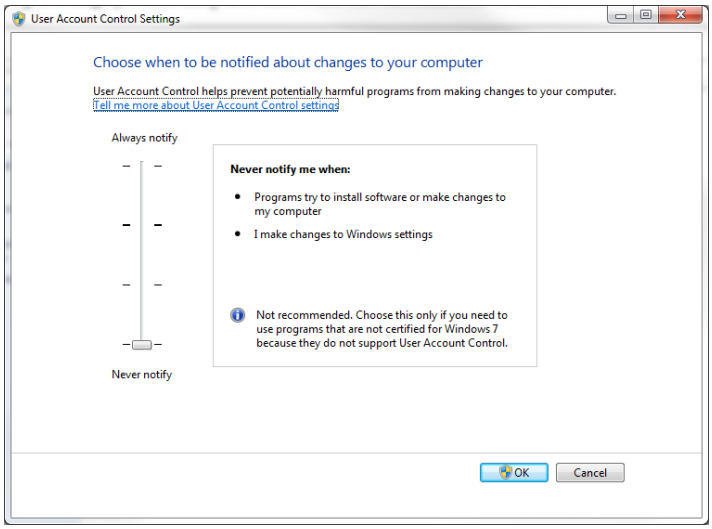
NOTE: You must have administrative privileges to install MechWorks PDM (Integrated and Standalone) and the Pre-requisite software.

Database

MechWorks PDM **does not** automatically install an instance of Microsoft SQL Server or Microsoft SQL Server Express; either of these must be installed prior to installing MechWorks PDM. MechWorks PDM does provide two database templates for creating a Microsoft SQL Server or Microsoft SQL Server Express database. A Microsoft Access database that allows for the "Upsizing" command to create the MechWorks PDM database within either Microsoft SQL Server or Microsoft SQL Server Express is available. Also available are MechWorks PDM_Eng.MDF and MechWorks PDM_Eng.LDF for performing a simple "Attach" command from within SQL Management Studio. For any installation that has X64 bit clients, Microsoft SQL Server or Microsoft SQL Server Express is required.

User Account Control (UAC)

Starting with Windows 7 the UAC does not necessarily need to be disabled for the installation of MechWorks PDM. It does need to be disabled for the installation of other services such as the DBWServer service (if installing the service locally on the workstation) or DBWAclServer service (on the server).



MechWorks PDM Registration Codes

Just as the CAD requires a registration code to function, MechWorks PDM (Integrated and Standalone) also requires a registration code. The registration code is known as the Serial Number. Typically, the Serial Numbers are delivered with the software (with the exception of Standalone). To obtain MechWorks PDM Serial Numbers, contact your software vendor.
Note: The relative SolidWorks serial number is required to obtain its related MechWorks PDM serial number.

NOTE: It is possible to run MechWorks PDM (Integrated and Standalone) without a serial number during its 30 day evaluation period.

Installation Methods

There are three basic methods of installations available for MechWorks PDM (Integrated and Standalone). These installation methods are:

Client – For all sequential installations after the first "Client + Server" installation: This installation will install MechWorks PDM (Integrated and Standalone) locally. It will also take advantage of having previously installed MechWorks PDM (Integrated and Standalone) as a Server Installation and modify the workstation's registry to appropriately connect to the mapped network location of the shared resource folders.

Individual – For use on a single workstation: This installation is typically used for installations that are not going to share a database and where users are not sharing data/files. (Not recommended for multi-user environments but will be used when UNC paths are used for the shared MECHWORKS_PDM_SERVER folder).

Client + Server – Install this once for each database site: This installation method should only be done once (excluding Distributed Sites). This installation process will configure the shared resource folders and requires a common shared network drive.*

Installation Type	Integrated MechWorks PDM	Standalone MechWorks PDM
Client	Yes	Yes
Individual	Yes	Yes
Client + Server (one time process)	Yes	Yes

***NOTE: MechWorks PDM (Integrated and Standalone) runs on the client's workstation. The application is always installed locally, even when selecting the Client+Server installation method. Using the Client+Server installation method allows the shared resources to be installed onto the selected shared path (referred to as the SERVER portion of the installation process). No part of the actual MechWorks PDM application runs from the server for a client's workstation.**

Benefits of Client+Server installation of MechWorks PDM

The benefits of performing a Client+Server installation of MechWorks PDM are many. The following lists some of the benefits:

- Users can share all of the common data (common resource files).
- One central location for customization resources.
- Windows 2008R2/2012/2012R2/2016 Security settings can be incorporated (ACL control).
- Some of the MechWorks PDM Advanced Options require a networked environment.
- Concurrent engineering functionality in MechWorks PDM, (example: MechWorks PDM Refresh).

Network Requirements*

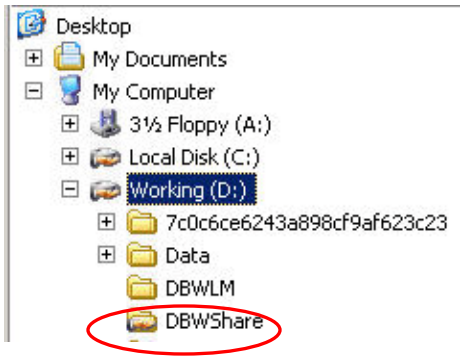
In order to share your MechWorks PDM resource files with other MechWorks PDM users, you will be required to have a networked environment. Typically, this is already the case. In the event that you do not have an established networked environment, here are a few requirements.

1. Shared drive to locate the common MechWorks PDM resource folders.
   - NOTE: If using the MechWorks PDM functionality of Local Mode, this mapped drive cannot be the same mapped drive as the physical data files that are to be managed by MechWorks PDM.
2. All computers that run MechWorks PDM (Integrated and Standalone) need to have the ability to connect to the designated shared drive.
3. For certain applications and advanced options, (like DBWAclServer), it is required that you create a Global Group for your MechWorks PDM users. **Note: DBWDomain is the name of a Global Group that your System Administrator will need to create and all MechWorks PDM users should be a member of this Global Group. The System Administrator can use a different Global Group name, but DBWDomain is the recommendation.**

Create the Shared folder on a Shared drive

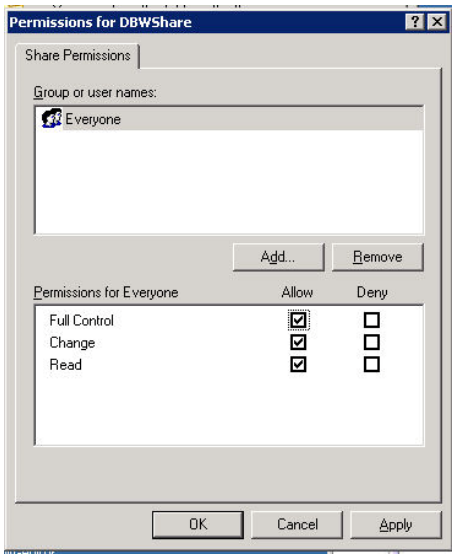
From the "network drive"

Prior to installing MechWorks PDM (Integrated and Standalone), you will need to establish a shared folder on a drive that all installations of MechWorks PDM (Integrated and Standalone) will have access to, your "server location". This is typically done by a System Administrator.



The name of this folder is not particular, but it is recommended that its name is something meaningful. In this example, **DBWShare** was used.

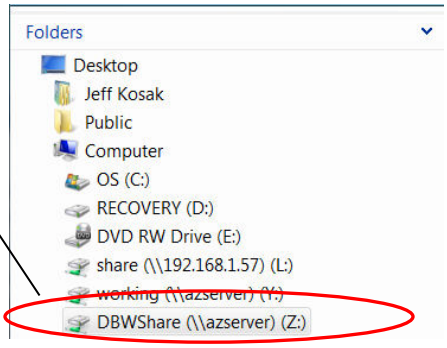
NOTE: The **Share Permissions** must be configured so users are able to write to the defined share, or the installation process will not copy the shared resource files to this drive!



From the "client" workstation

With the shared folder established, you need to map that folder from the "client" workstation. When mapping the drive, it is recommended that you choose a common mapping scheme for each workstation.

This is the mapped drive on the "client" workstation. In this example, "**Z**:" is the mapped drive letter. Any available drive mapping letter can be used, but it is recommended that all "client" workstations consistently use the same drive mapping. This is referenced as the MechWorks PDM shared folder.



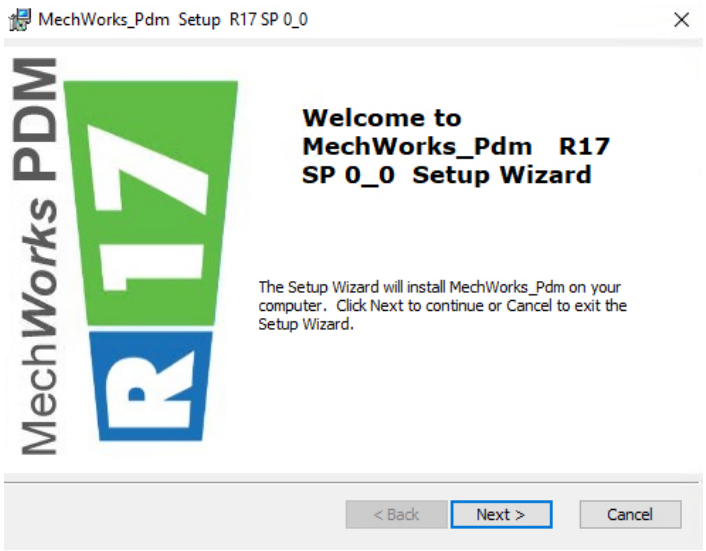
Integrated and Standalone Installation Process (Client + Server)

Prior to installing MechWorks PDM on any workstations, you should ensure that the Operating System is up-to-date with all available Microsoft patches (Microsoft Windows Updates).

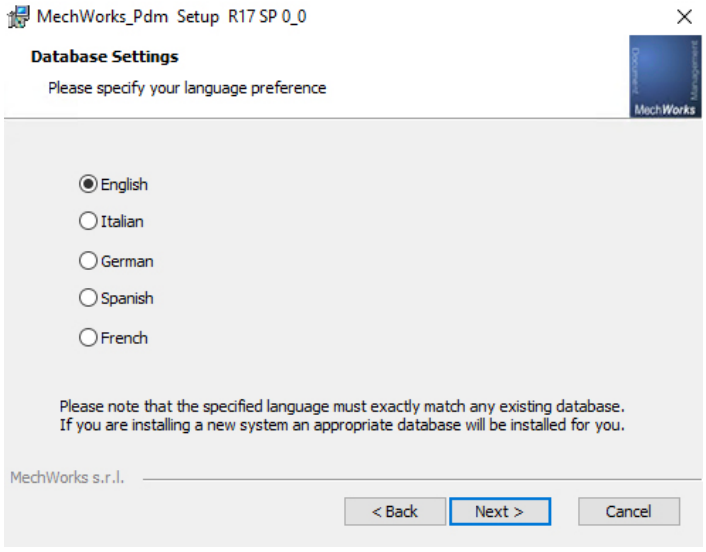
MechWorks PDM Client+Server Installation:

NOTE: You must have the appropriate Administrative permissions on the workstation, to be able to install MechWorks PDM.

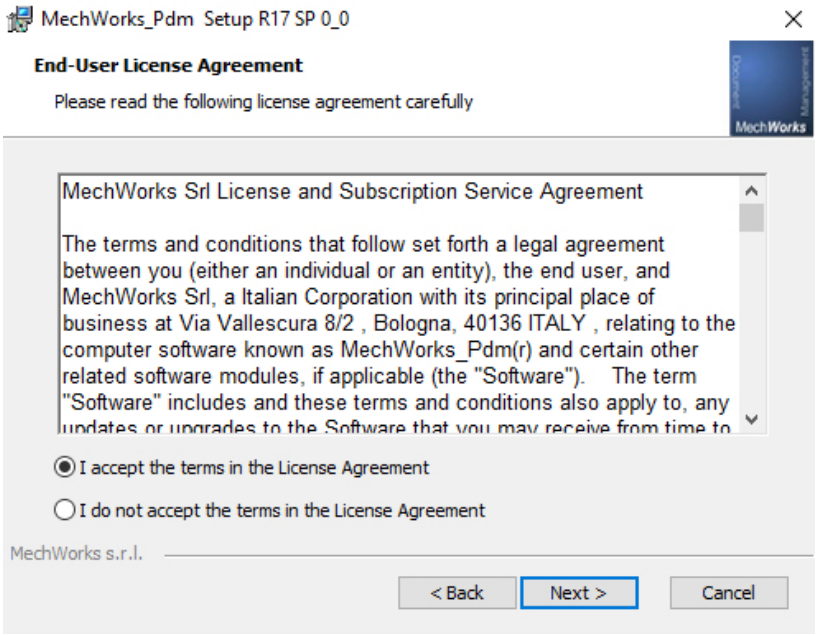
1. To install MechWorks PDM, download the MechWorks_Pdm_Msi_xxx.exe file from the www.mechworks.com web site. Double mouse click the downloaded MechWorks_Pdm_Msi_xxx.exe file to begin the installation process. Select the [**Next]** button to continue the installation.



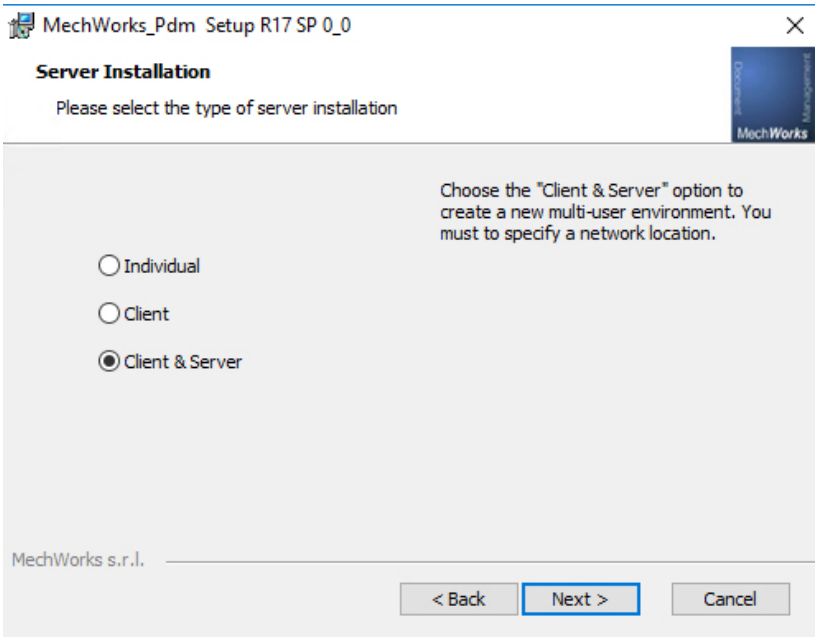
2. The first screen in the installation wizard is the Regional Settings screen. In this screen, select the language you wish to install and then select the [**Next]** button.



3. Following the Regional Setting is the End-User License Agreement. To continue with the installation, this license agreement must be accepted. Select the **I accept the terms in the License Agreement** option and select the [**Next]** button.



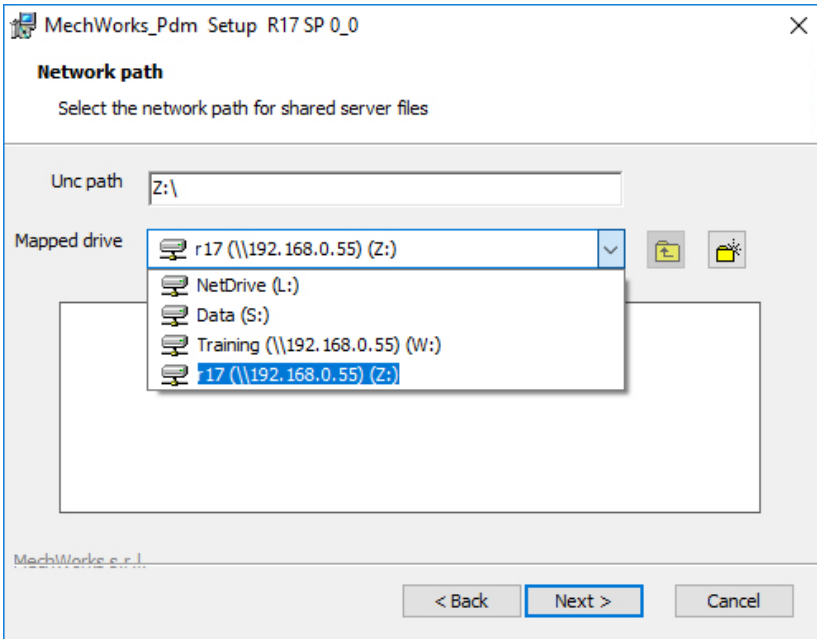
4. You will be prompted for the type of installation method to be performed. In this example, we will be using the **Client & Server** type of install. Select the **Client & Server** type of installation and select the [**Next]** button. This will begin the installation process and you will be prompted to answer a few questions throughout the install process.



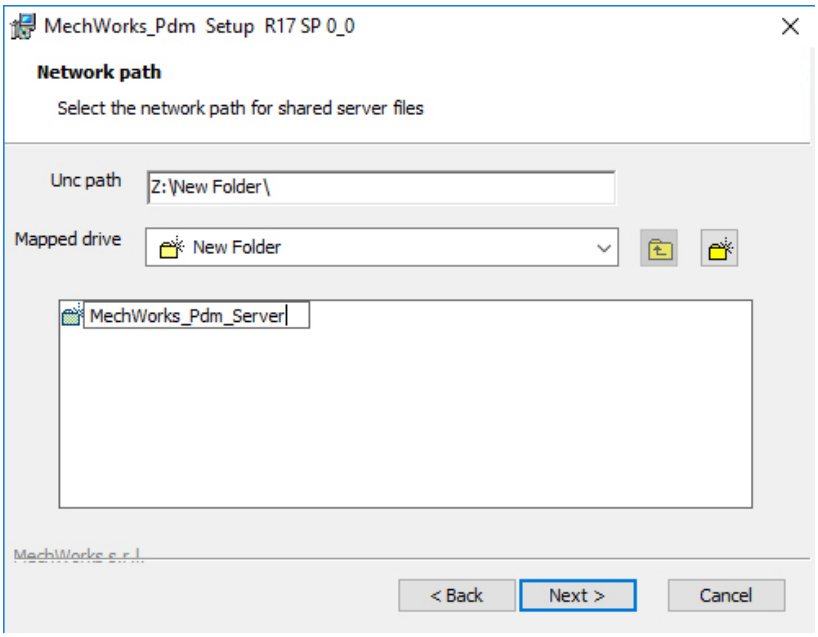
Installation Methods

There are three types of installation methods available for MechWorks PDM (**Client**, **Individual**, and **Client + Server**). The type of installation that you choose is determined by whether you are using a Network installation or a Local installation. For this example, we will continue with the **Client+Server** method of installation. The other two methods of installations will be covered in the lessons that follow.

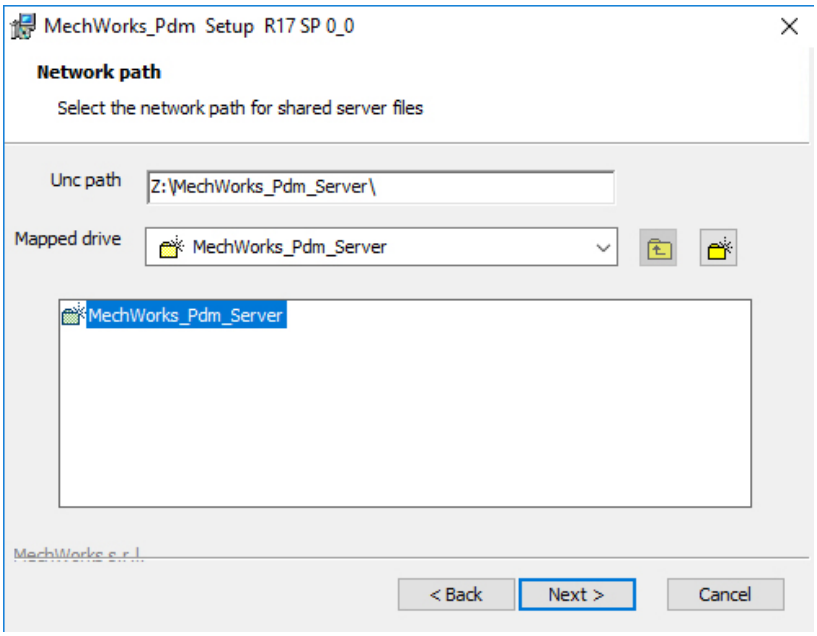
5. The next screen presented is the **Network Location**. This is the shared network drive on which the shared resource files will be installed. From the **Look in:** drop down list, select the network drive (this is the commonly mapped drive that all MechWorks PDM users will need to have available to them) on which the common resource files should be installed onto.



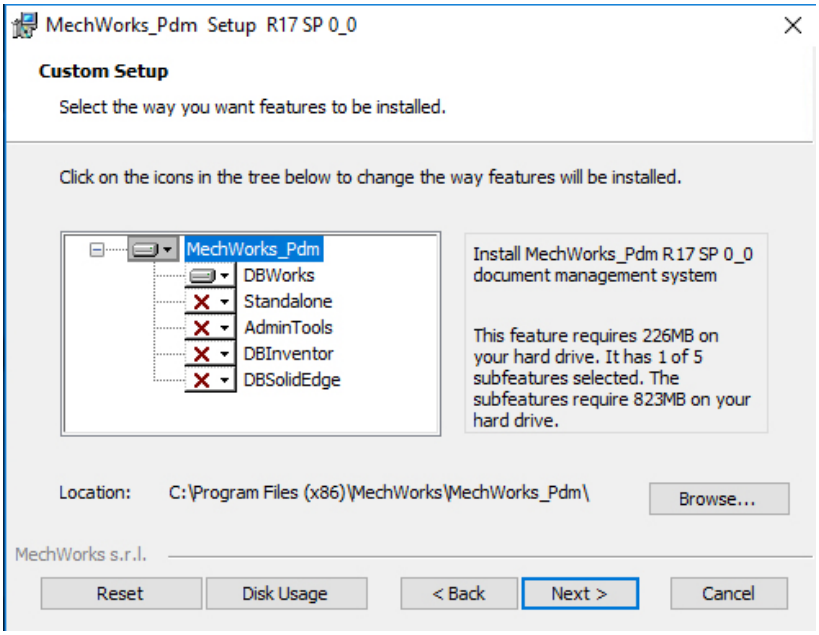
6. If the MECHWORKS PDM_SERVER folder does not exist it may need to be created. Select the New Folder button to create the MECHWORKS PDM_SERVER folder. This is the folder that will contain the Shared Resources for MechWorks PDM.



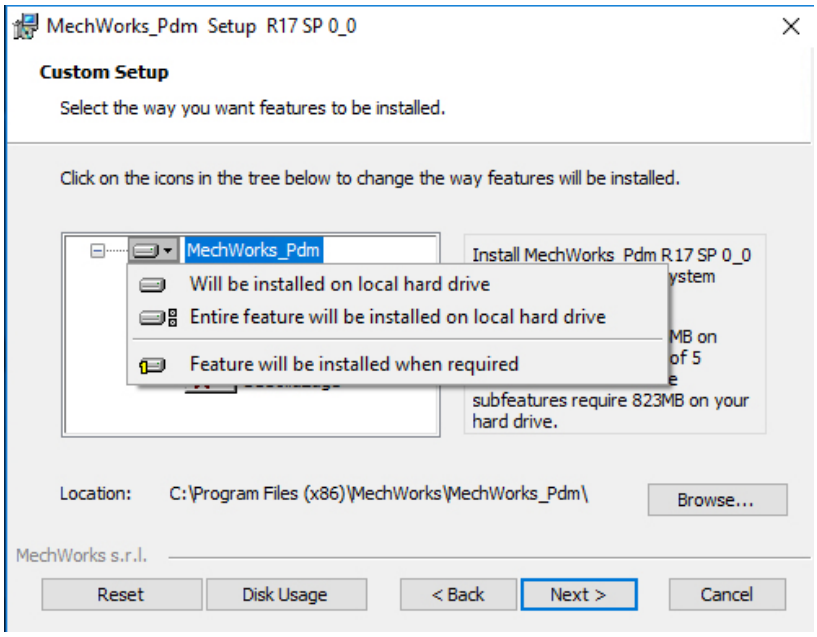
7. Select the newly created folder. Select the [**Next]** button to continue.



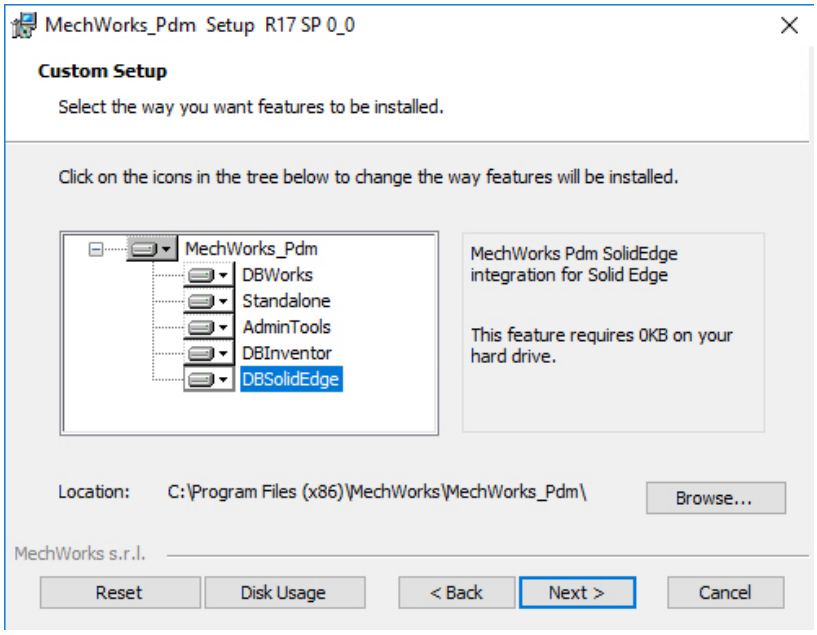
8. Following the Network Location screen is the **Custom Setup** screen. Using this screen, select the specific applications that are needed for the current workstation. All corresponding CAD applications that are installed are selected by default. Optional applications are the Standalone application as well as the Admin Tools.



9. To install all applications (Integrated, Standalone and the Admin tools), select the drop down button on the MechWorks_PDM icon and select the **Entire feature will be installed on local hard drive** command.



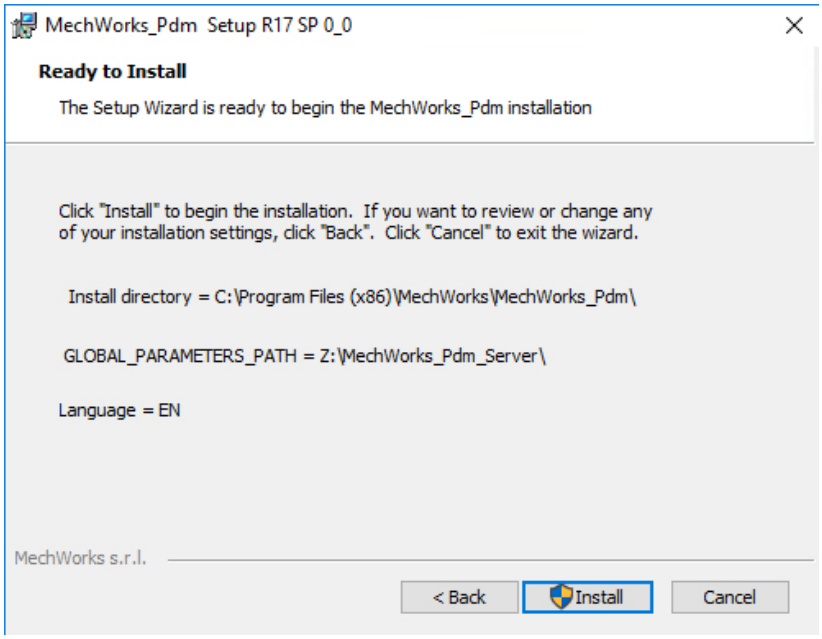
10. All applications should now be displayed without the red X (indicating that they will be installed). Select the **[Next]** button to continue. NOTE: If an application's icon is displayed with the red X next to it, then it is not going to be installed.



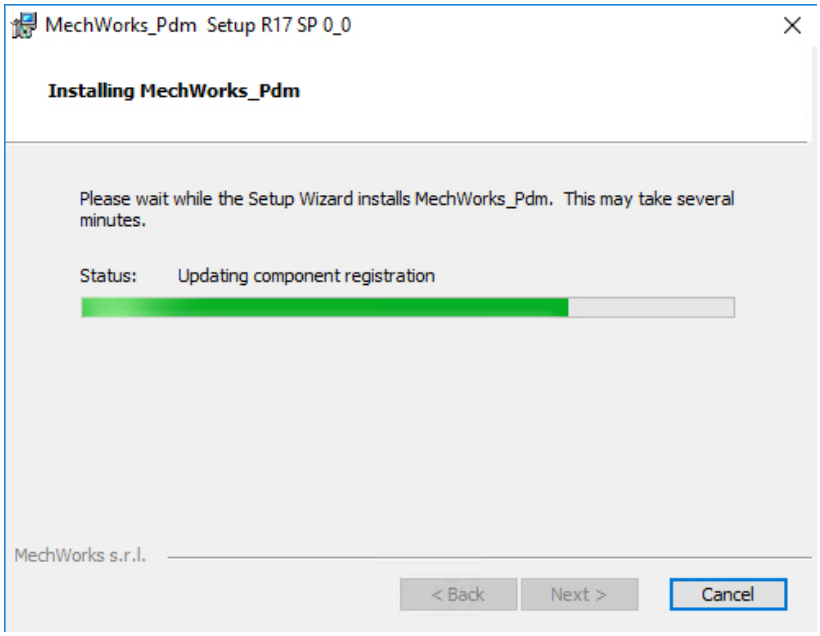
NOTE: Admin Tools

MechWorks PDM has a number of Administration Tools that are available to install. These tools include **DBCustomizer**, **Material Editor**, **Custom Property Manager**, **Category Maker**, **Scripting**, etc…. These tools will help with the administration of MechWorks PDM and they can be installed on any or all of the workstations. It is a common practice to only install these tools on the workstations used by the PDM Administrator. The Admin Tools are explained in the lessons that follow.

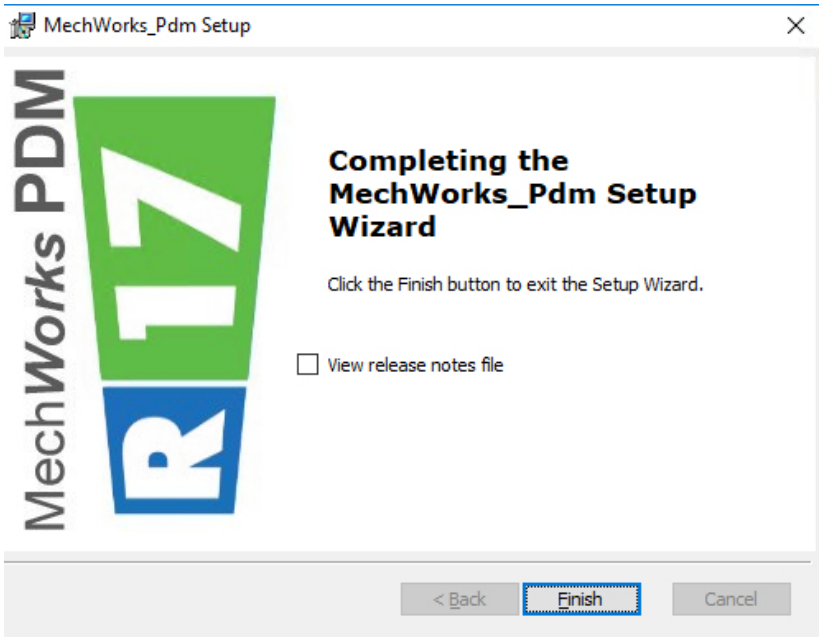
11. You are now ready to install. If all settings are correct, select the [**Install**] button.



12. During the installation process, a Status Bar will be displayed to display the status of the installation process.



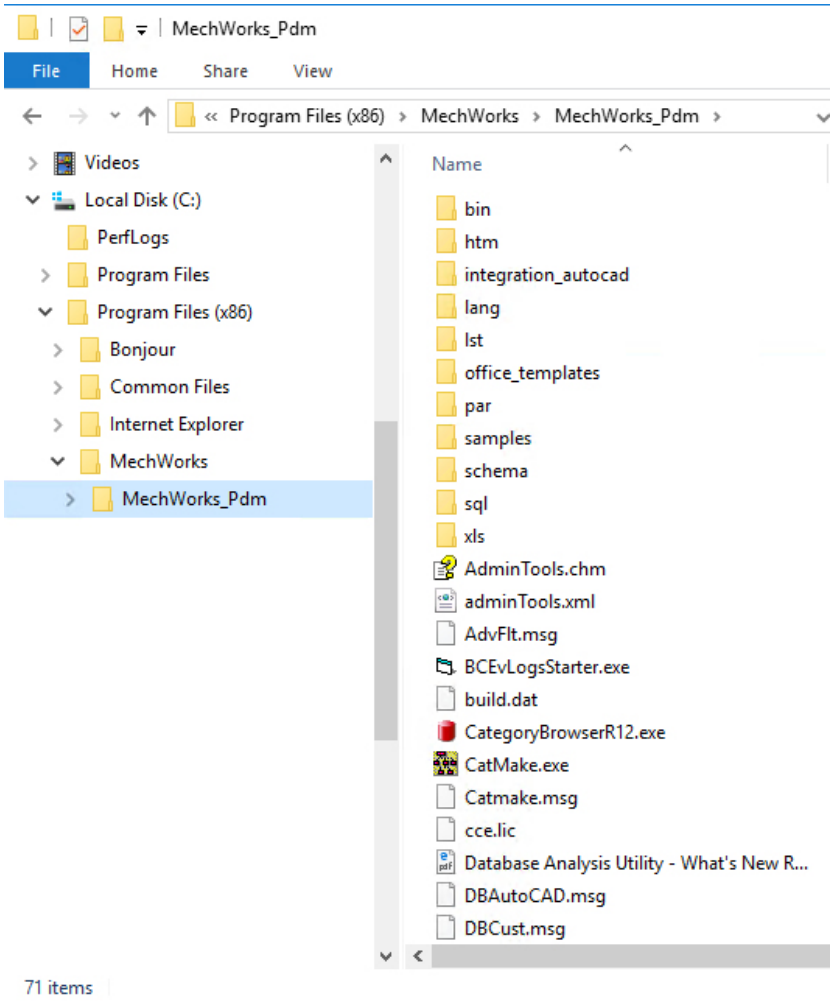
13. Once complete, the **Completing the MechWorks_PDM Setup Wizard** screen will be displayed. Select its **[Finish]** button. **Note: In some cases, a reboot of the workstation may be required to finish the entire installation process.**



What has been installed on the Workstation?

Now that the installation is complete, what exactly has been installed and where? Continuing with this lesson's example, both MechWorks PDM **Integrated** and MechWorks PDM **Standalone** were installed using the Client+Server method of installation. The applications (MechWorks PDM Integrated and MechWorks PDM Standalone) have been installed locally on the workstation. This includes the entire application as well as the Administrative Tools (provided that they were selected during the installation process).

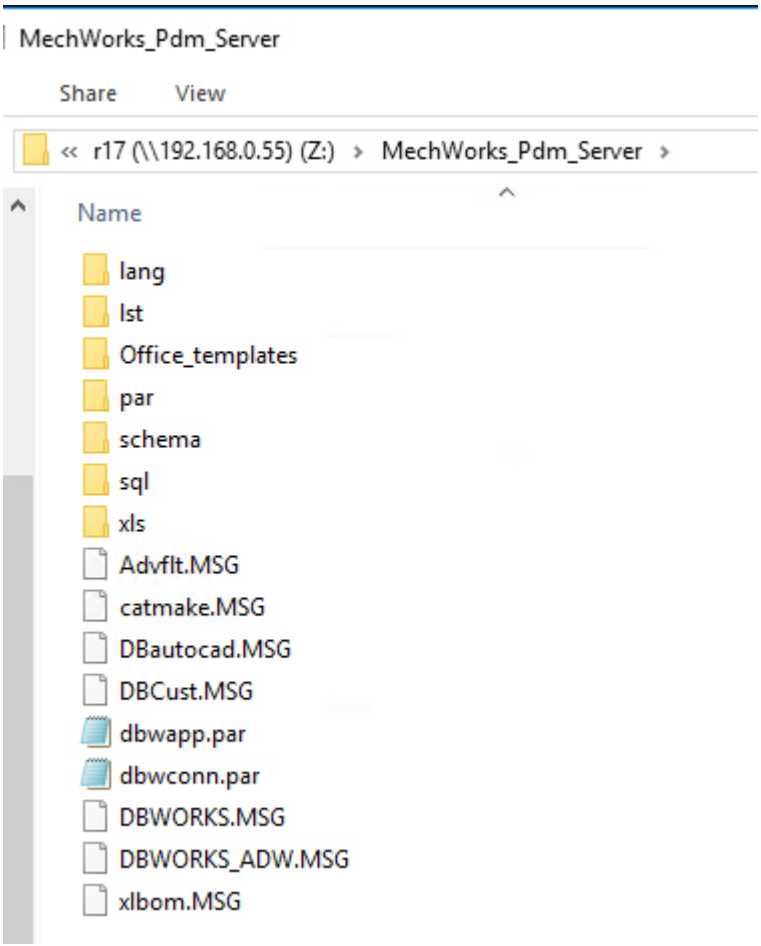
Local Workstation



By default, MechWorks PDM Integrated, DBInventor Integrated, DBSolidEdge Integrated and Standalone are all installed within the **C:\Program Files (x86)\MechWorks\MechWorks PDM** folder on a 64 bit workstation and within the **C:\Program Files\MechWorks\MechWorks PDM** folder on a 32 bit workstation.

What has been installed on the Server (shared drive)?

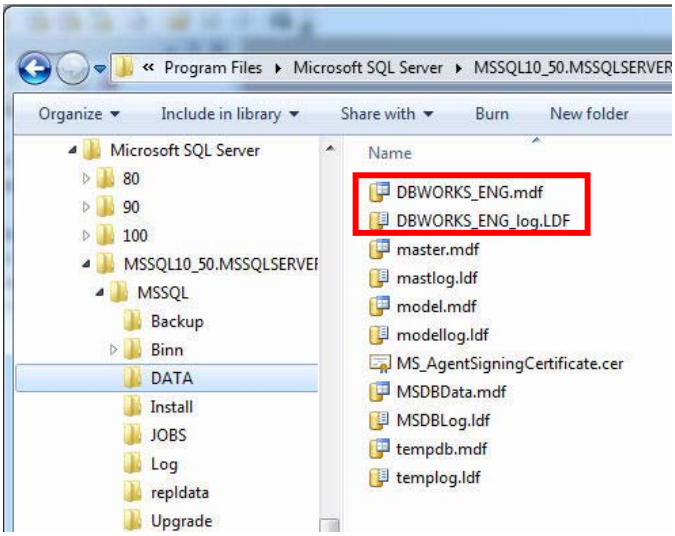
On the server, the shared resource files have been copied into the MechWorks_Pdm_Server folder.



The figure above shows the folders that have been automatically created within the selected **Z:\MechWorks_Pdm_Server** folder.

Attaching the MECHWORKS PDM_ENG database in SQL Server

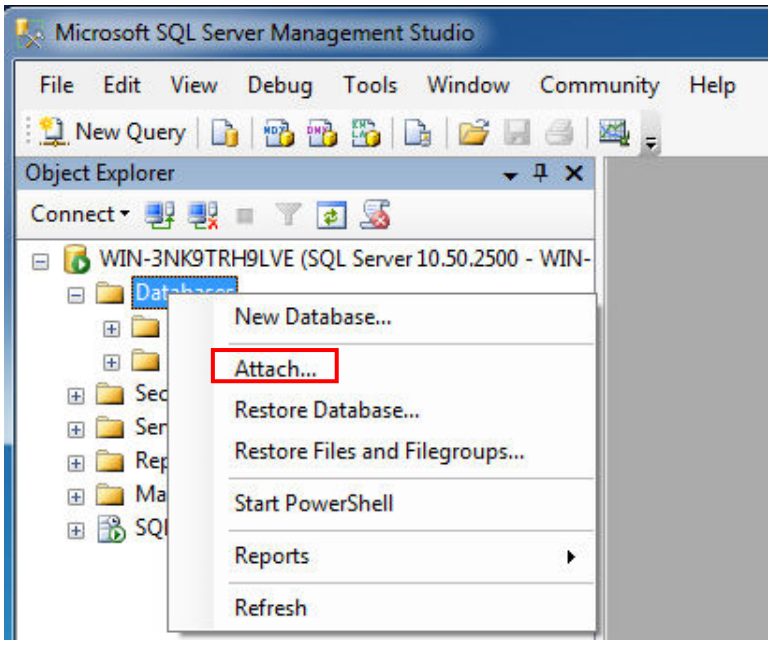
To take advantage of the provided copy of the DBWORKS_ENG SQL Server database, first locate the two files, **DBWORKS_ENG.mdf** and **DBWORKS_ENG_log.ldf**. Copy these two files to the data folder for SQL Server. This location is based on what has been defined during the installation of SQL Server but the default location is:
C:\Program Files\Microsoft SQLServer\MSSQL10_50.MSSQLSERVER\MSSQL\DATA



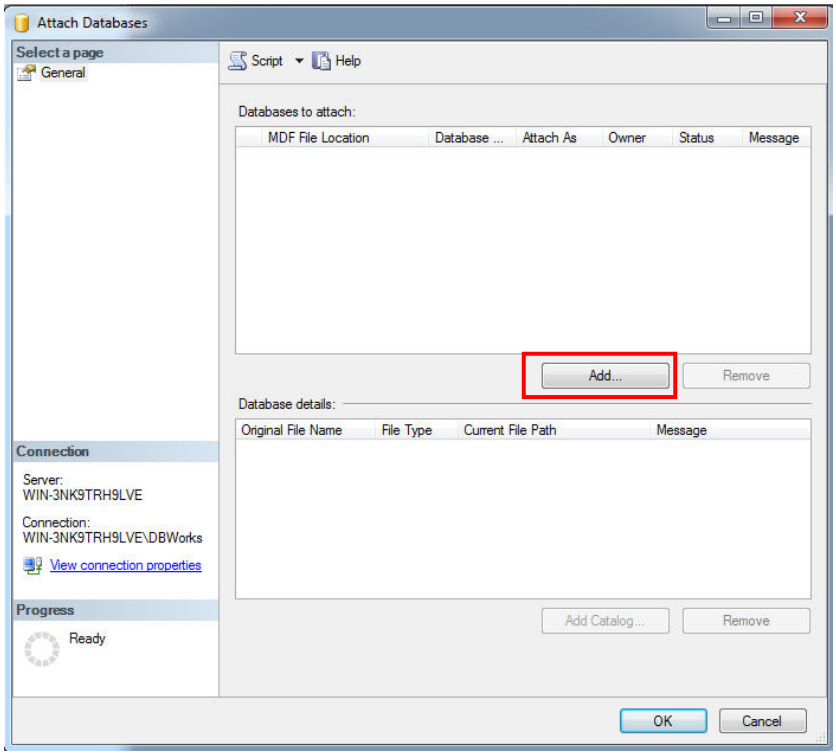
Note: The DBWORKS_ENG.mdf database is a SQL Server 2008 R2 database and can only be used with either SQL Server 2008 R2/2012/2014/2016 or SQL Server 2008 Express R2/2012/2014/2016.

After the files have been copied…

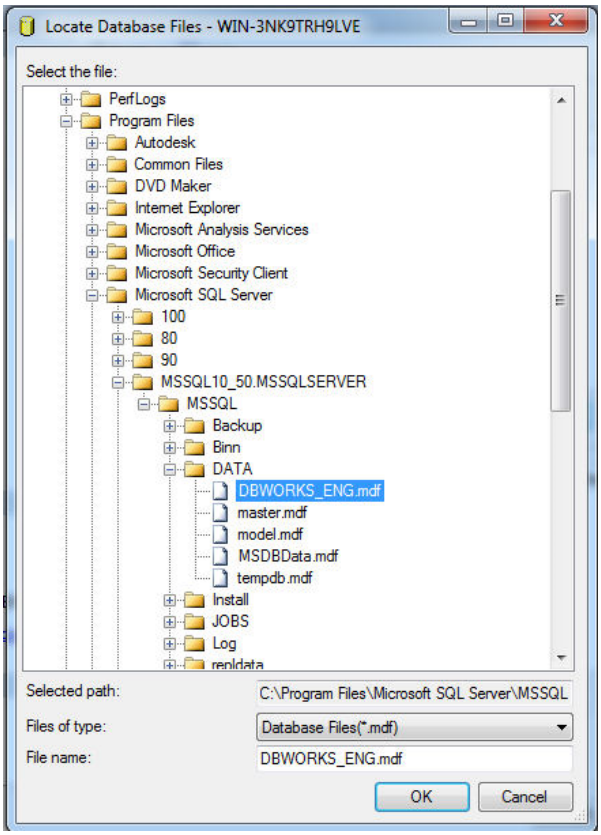
1. Open Microsoft SQL Server Management Studio. To attach the database, right mouse select the **Database** and select the **Attach…** command.



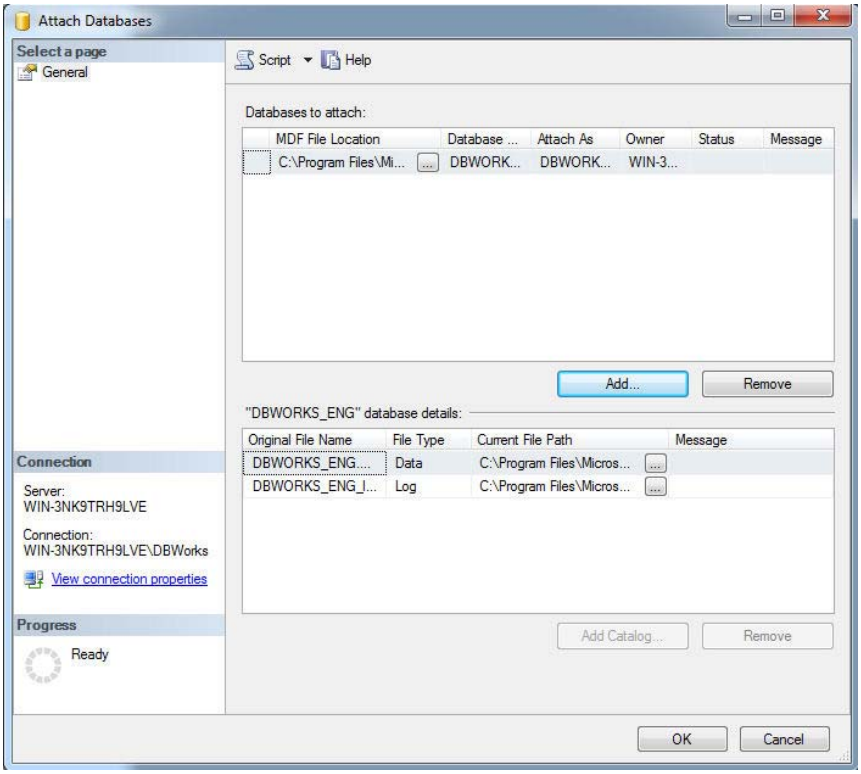
2. In the Attach Database dialog choose the **Add…** button.



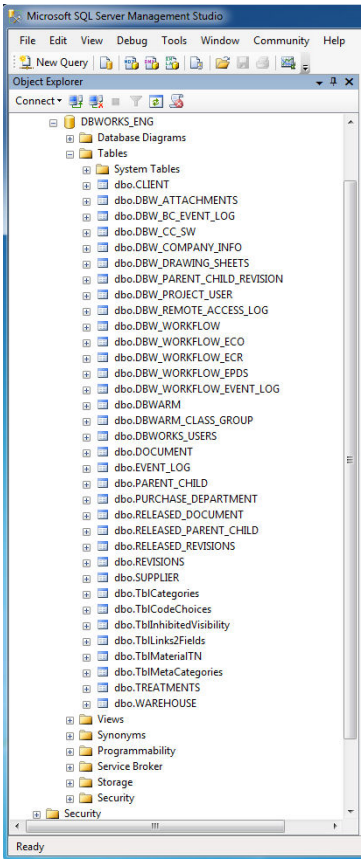
3. In the **Locate Database Files** dialog, locate and select the MechWorks PDM database that is going to be attached. If the provided database is being used then select the **MECHWORKS PDM_ENG.mdf** database file. Choose the **OK** button to continue.



4. The **Attach Database** dialog will now display the details for the database. To finish the attaching of the database select the **OK** button.



5. The **DBWORKS_ENG** database is now attached in SQL Server and is available to be used for MechWorks PDM.



Understanding MechWorks PDM (Integrated and Standalone) data sources

MechWorks PDM (Integrated and Standalone) requires a standard ODBC data source connection to the DBWORKS database, but if the advanced feature of **Released Database** mode is utilized, a second data source will be required. The data sources are:

DBWORKS

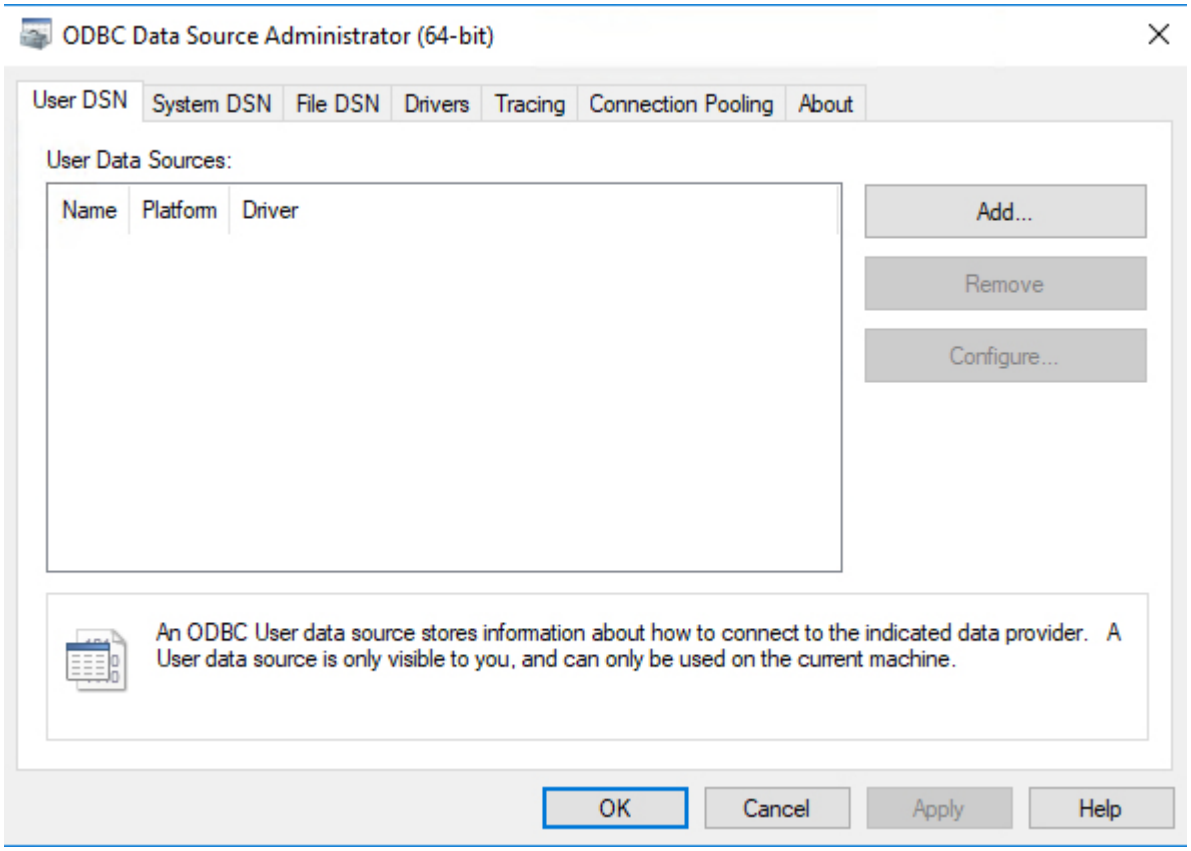
This is the primary data source that identifies the database that contains all of the core information about an organization's SolidWorks®\Inventor®\Solid Edge® and generic document records. Within a shared network environment, all workstations must share the same (common) DBWORKS data source to be able to work on the same files/data. This is the database that will contain the majority of the metadata that a user provides and distributes throughout the organization.

DBWORKS_BRIEFCASE

This is an internal resource data source used by MechWorks PDM, and it should not be modified.

Open Database Connectivity (ODBC)

MechWorks PDM (Integrated and Standalone) uses Open Database Connectivity, commonly known as ODBC, to access data from a variety of databases. The ODBC Data Source Administrator utility lists all the locally available data sources that can be accessed from different programs at the same time and therefore must be considered a system resource for data input/output. ODBC is an industry standard middleware application.



Accessing the Data Sources

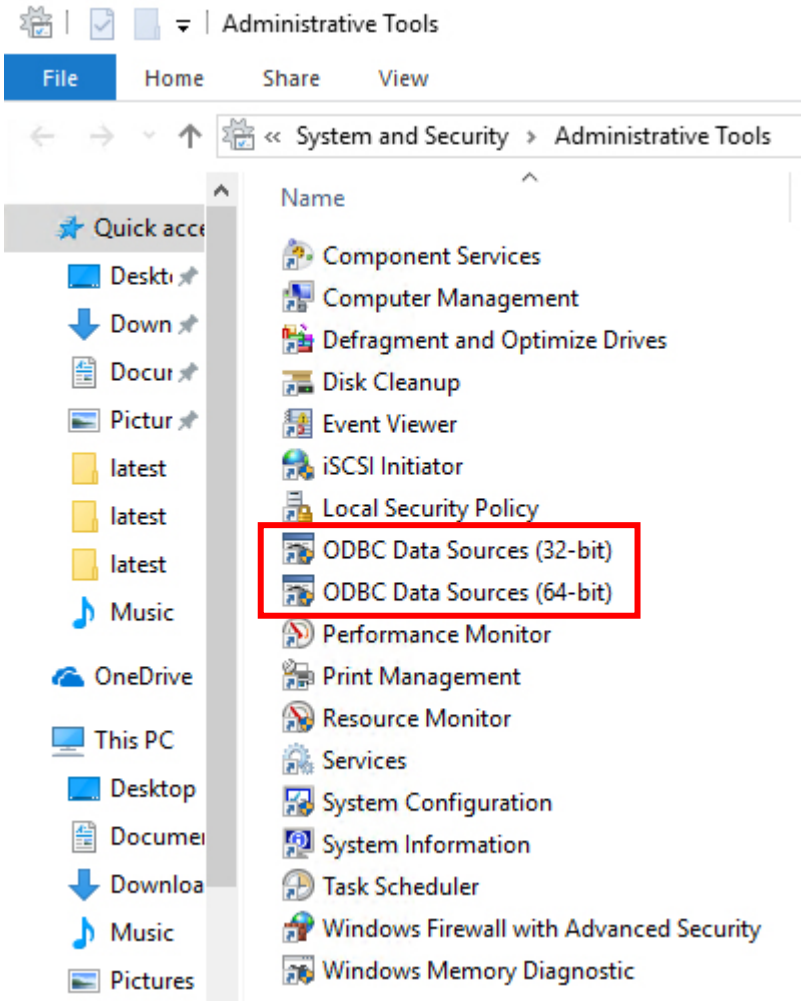
There are times when you will need to validate your ODBC connections. You may need to make changes to the ODBC paths, (example, more than one user will be accessing the same workstation throughout a 24 hour timeframe and/or you are going to switch between a production and test environment). To access the ODBC controls, you will run the **Data Sources'** manager. Depending on the version of Windows that is installed (32bit vs. 64bit), there are different methods for accessing the **Data Sources'** Administrative utility. This section will cover ODBC settings for a Windows 64bit workstation.

Prior to starting MechWorks PDM, it is recommended to establish/configure the ODBC connections that are required for MechWorks PDM to work correctly. The connection that is mandatory for proper functionality within MechWorks PDM is the **DBWORKS** data source.

Windows 64 bit Operating Systems

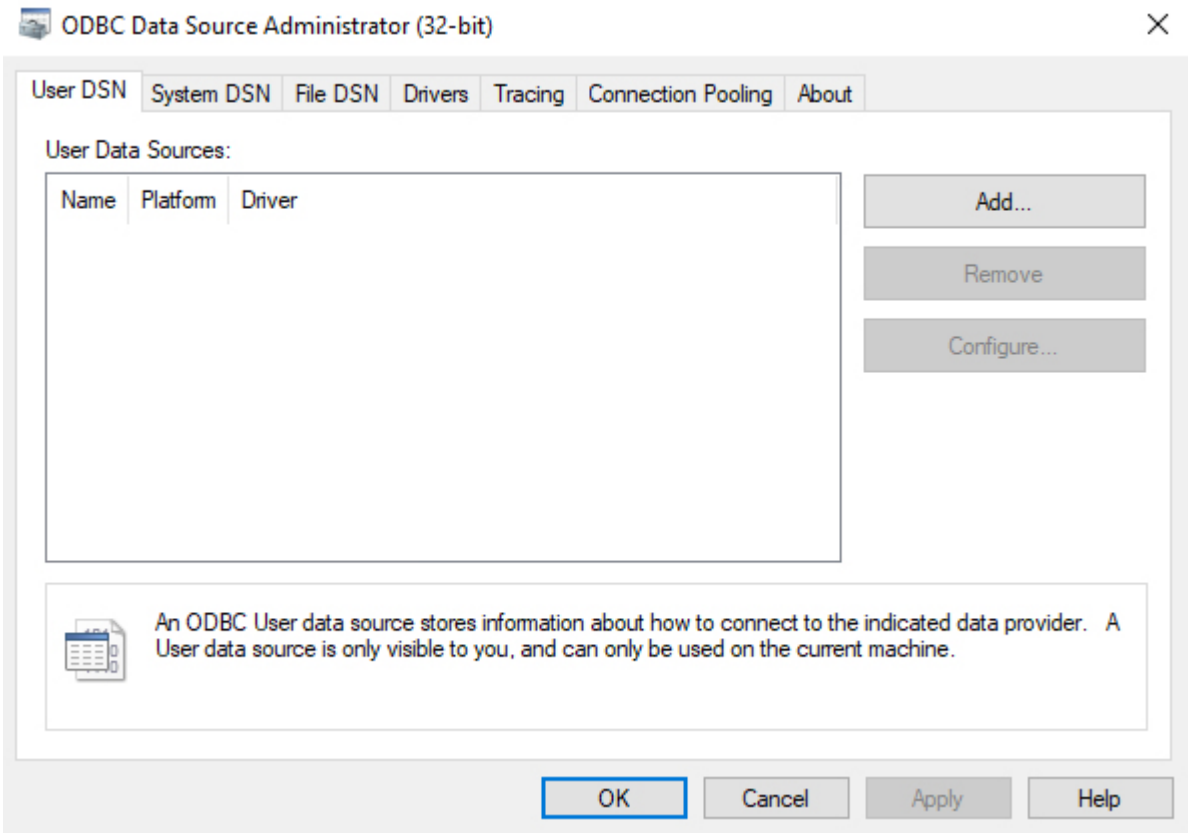
*To **CREATE** the ODBC data source connection for a SQL Server database.*

On Windows 7/8 workstations, MechWorks PDM provides an ODBC manager for both X32 and X64 systems directly in the **MechWorks PDM** program folder. On Windows 10 machines, the ODBC Data Sources can be access through the Control Panel -> Administrator Tools.



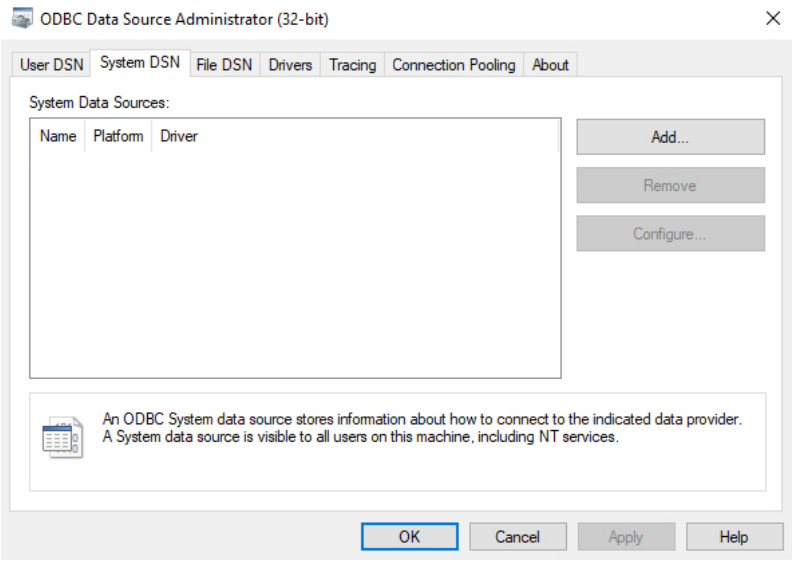
1. On a Windows 10 workstation, navigate to the **Control Panel -> System and Security-> Administrative Tools**. To set the ODBC settings both the **ODBC Data Sources (32-bit)** and **ODBC Data Sources (64-bit)** commands will be utilized. Because MechWorks PDM utilizes some 32-bit functions, a connection to the database must be established for both 32-bit and 64-bit.

2. To establish the ODBC connection for 32-bit, locate the **ODBC Data Sources (32-bit)** command and double mouse click it. This will open the ODBC Data Source Administrator specifically for the x32 ODBC connection.



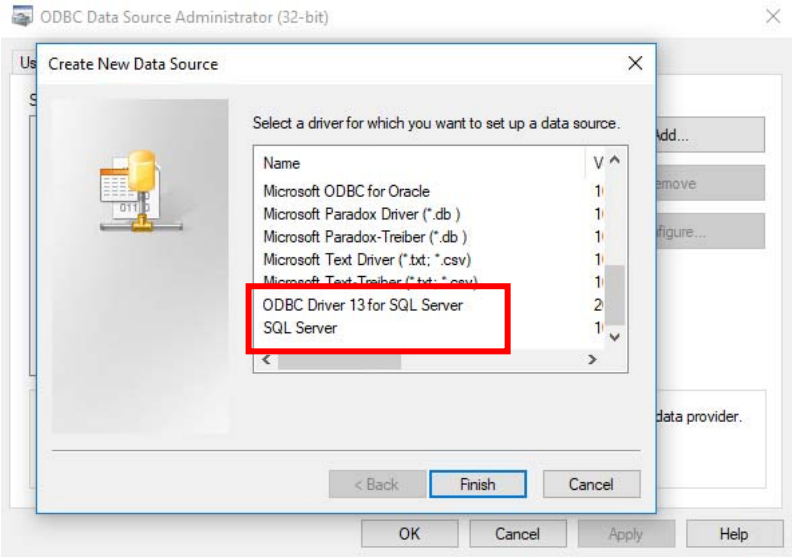
The **ODBC Data Source Administrator** dialog.

3. To ensure that the ODBC settings are correct (no matter who logs onto the workstation) change to the **System DSN** tab. Select the **[Add]** button to add the data source. ***Caution: Do NOT create a data source within the System DSN tab with the same exact name as a data source within the User DSN tab.***

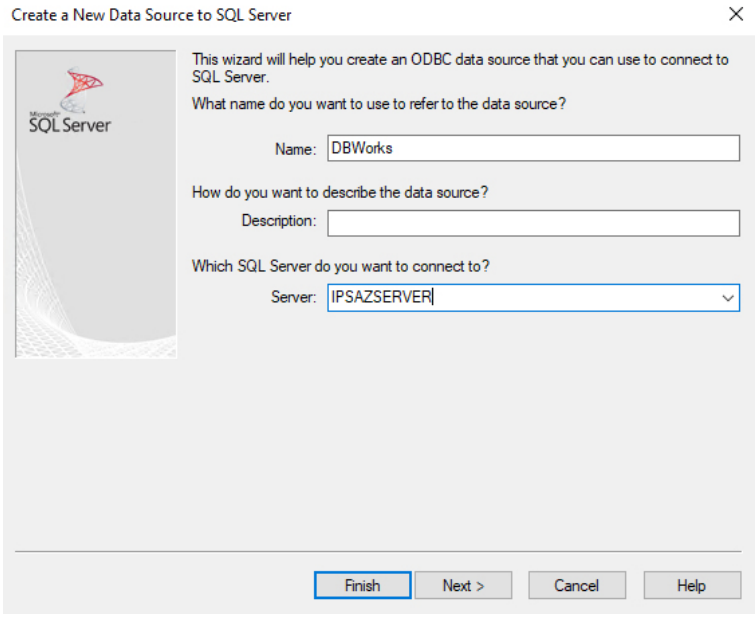


MechWorks PDM support two ODBC drivers for SQL Server, **ODBC Driver 13 for SQL Server** and **SQL Server**. Typically, it is recommended to utilize the **ODBC Driver 13 for SQL Server**.

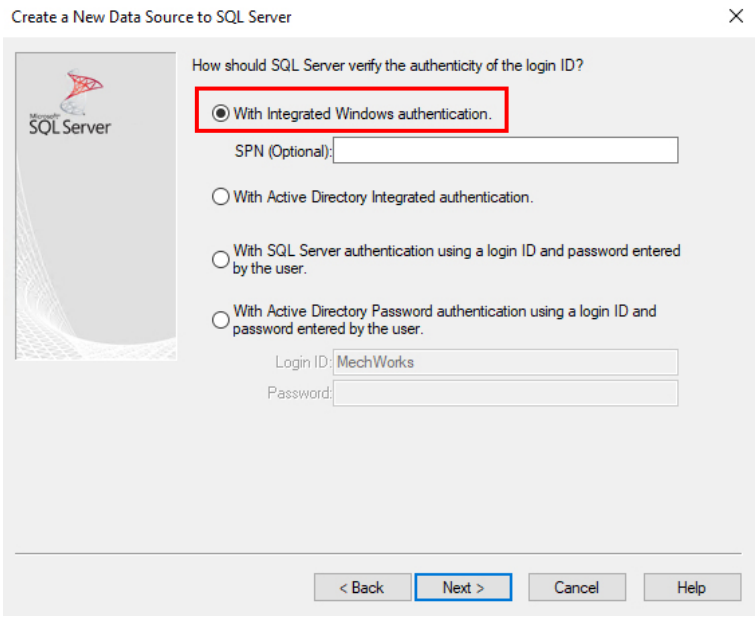
4. Select **ODBC Driver 13 for SQL Server** from the list of available data source drivers and then select the **[Finish]** button.



5. Enter the Data Source Name: **DBWORKS**. If desired, an optional description of this data source can be entered in the Description area. Select the **[Server]** drop down list and choose the SQL Server or type in the server name. Select the **[Next]** button to continue.

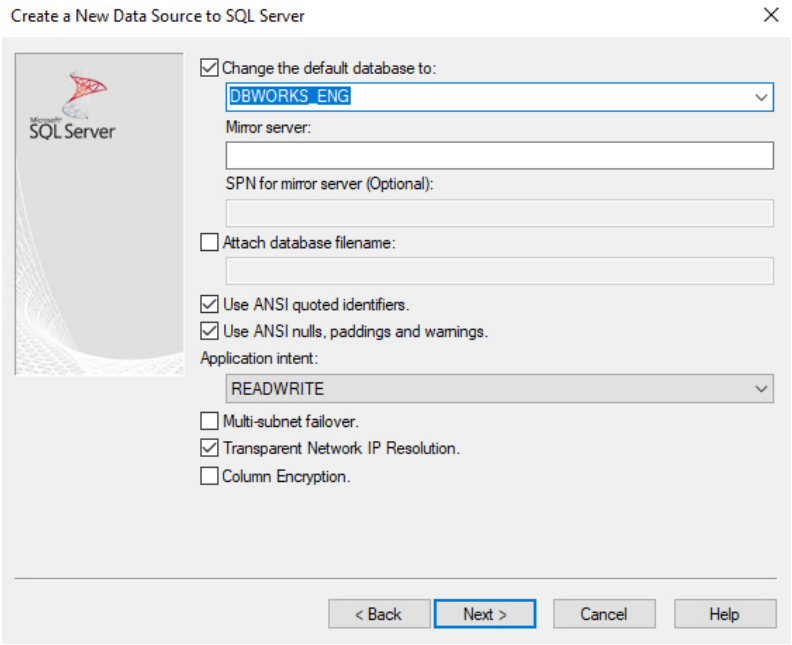


6. The next screen is the authentication screen. For standard MechWorks PDM installation, use the **With Windows NT authentication using the network login ID** option. This is the standard procedure since we recommend that all MechWorks PDM users are assigned to a Global Group (example: **DBWDomain**) and that this group has previously been granted the proper permissions to the upsized MechWorks PDM database. It is critical that the assignments within this authentication dialog are correct before proceeding. Select the **[Next]** button to continue.

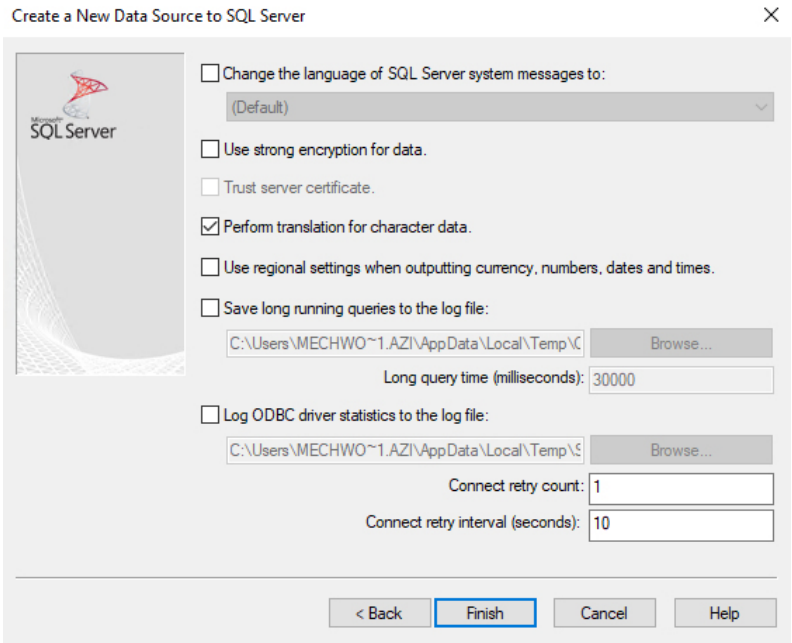


NOTE: Although SQL authentication can be used; it does requires an additional update to the **DBWCONN.par** file to start MechWorks PDM correctly. The **DBWCONN.par** file is a special resource file used by the MechWorks PDM (Integrated and Standalone) applications and it is located in the installation folder of MechWorks PDM (Integrated and Standalone).

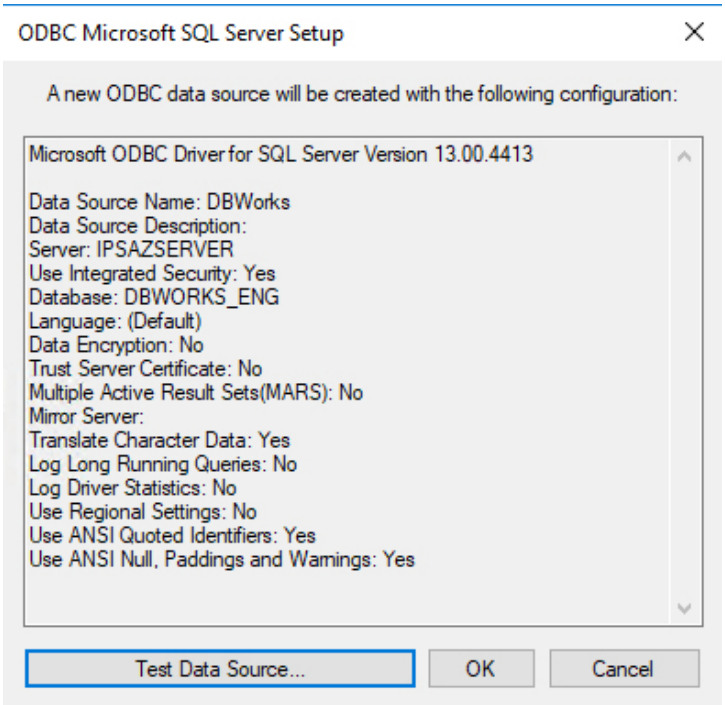
7. The next dialog requires that the SQL Database is selected. After selecting the **DBWORKS_ENG** (in this example the database name is DBWORKS_ENG but it could be named DBWORKS) database from the drop down list, be sure to activate the **Change the default database to:** box. Select the **[Next]** button to continue.



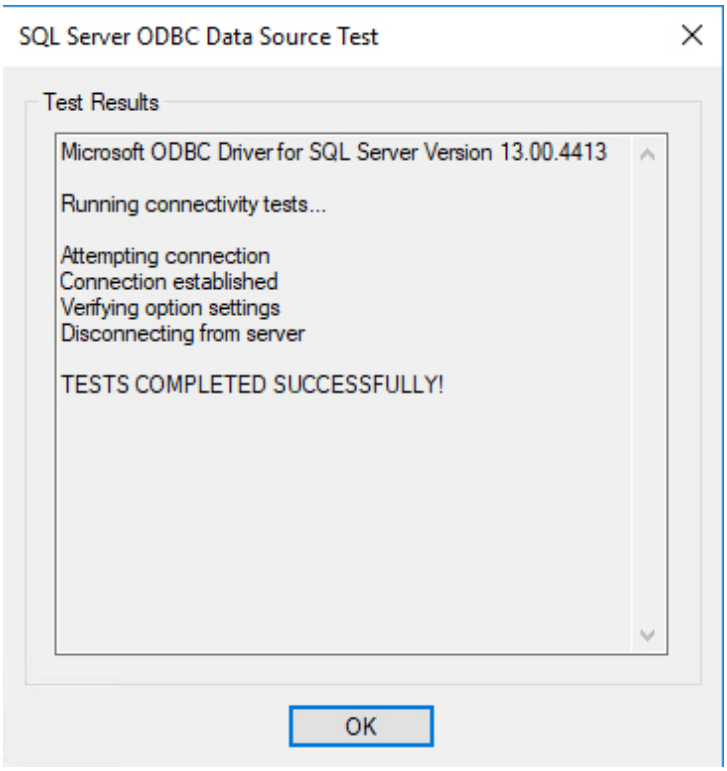
8. The final configuration screen does not require any changes, accept its default settings. Select the **[Finish]** button to allow for the testing of the connection.

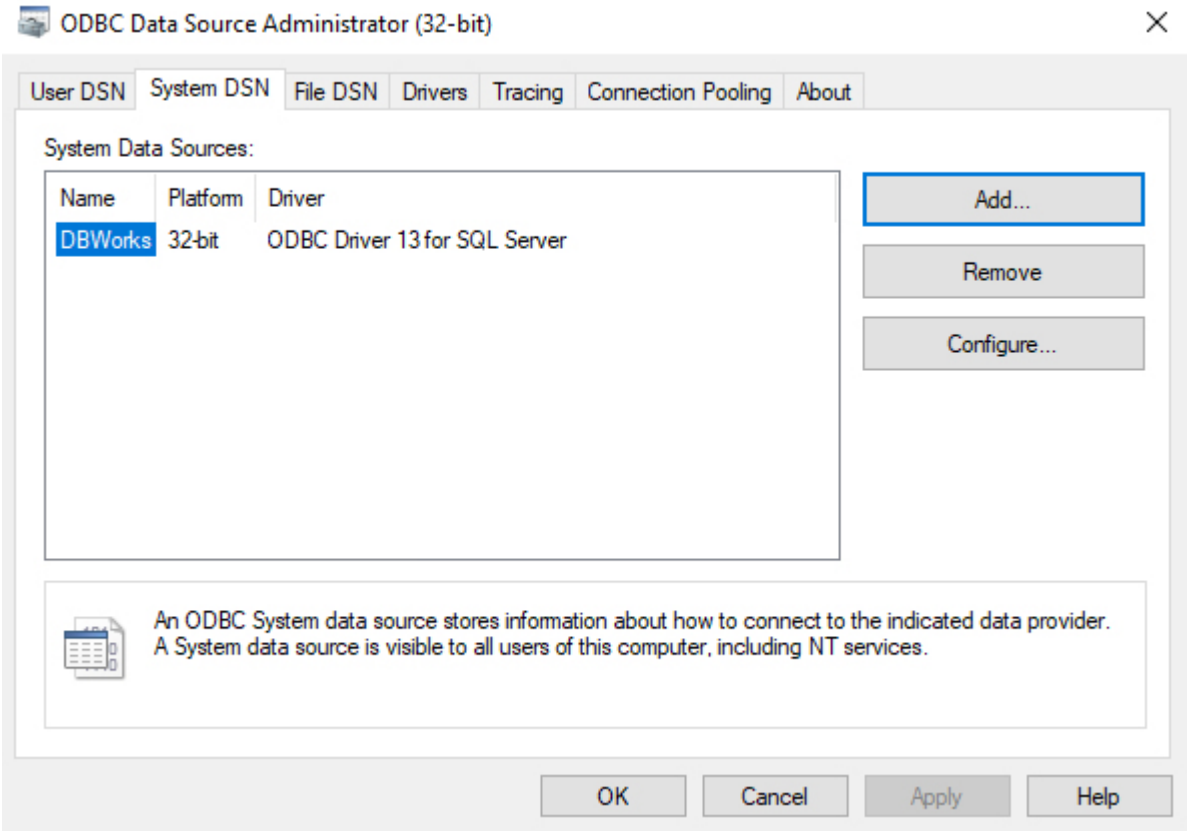


10. From the review screen, validate that the displayed settings are correct. If the settings look correct, select the [**Test Data Source…]** button. If changes are required, select the [**Cancel]** button and begin the process again.



11. After selecting the [**Test Data Source…]** button, review the results of the test. The test must complete successfully for a qualified data source connection. If the test fails, changes are required. Select the [**OK]** button when done.

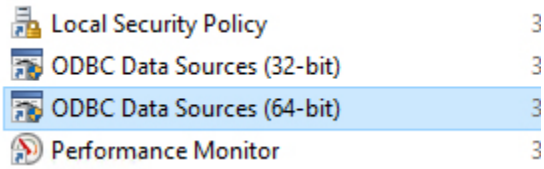


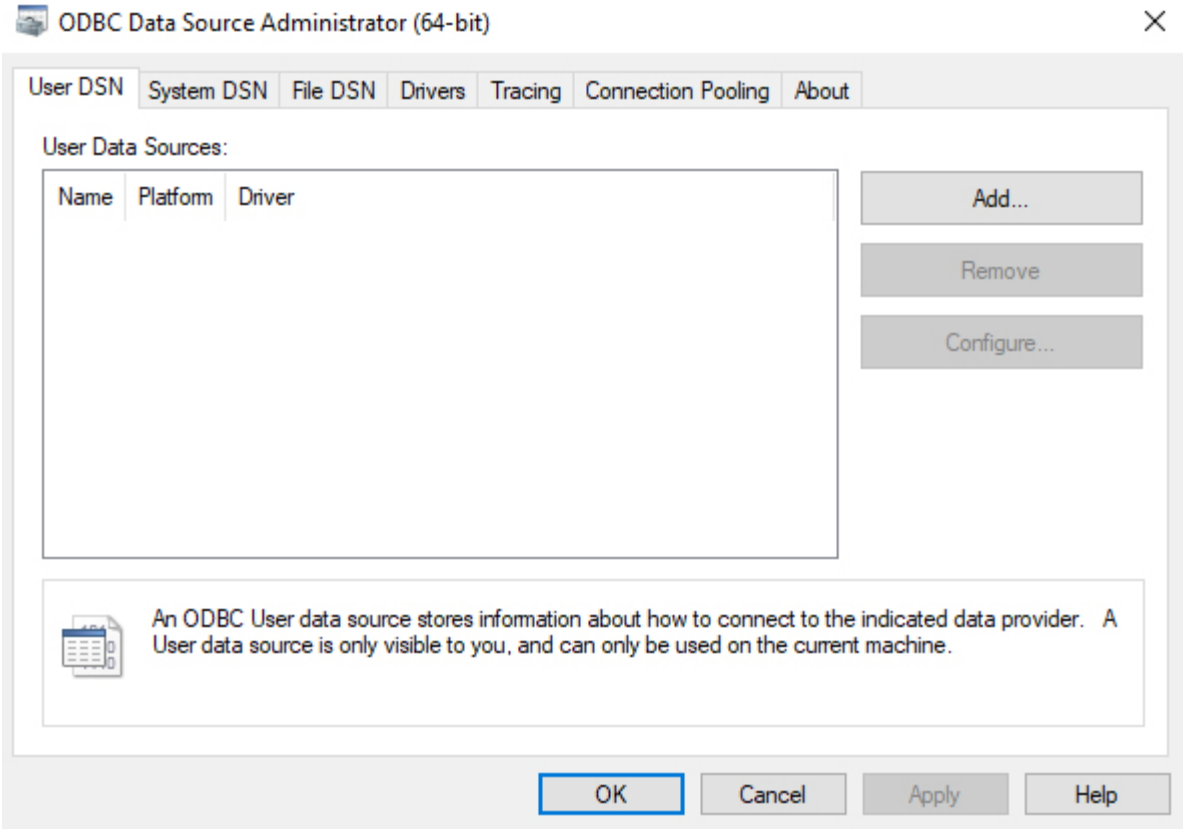


The DBWORKS SQL Server data source is now established for MechWorks PDM (Integrated and Standalone).

With 32-bit ODBC Data Source created, create the 64-bit ODBC Data Source.

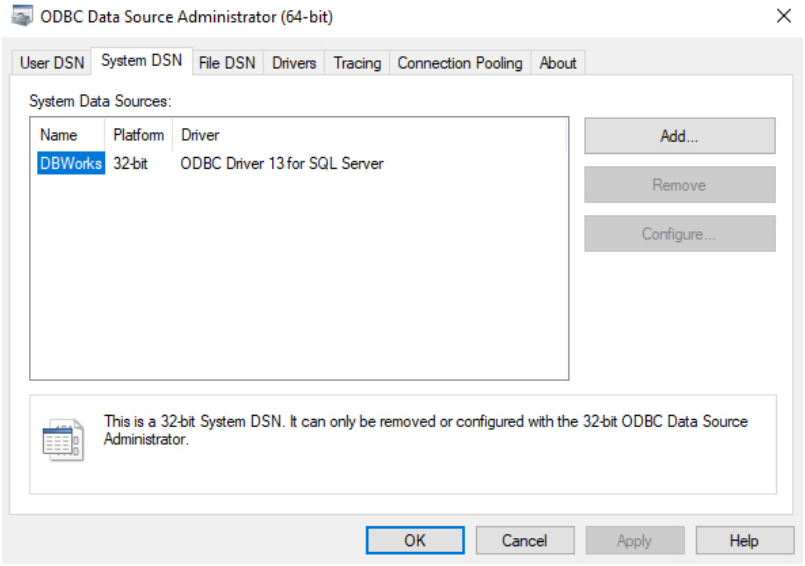
1. Locate the **ODBC Data Sources (64-bit)** command from within the Control Panel -> System and Security -> Administrative Tools and double mouse click it.



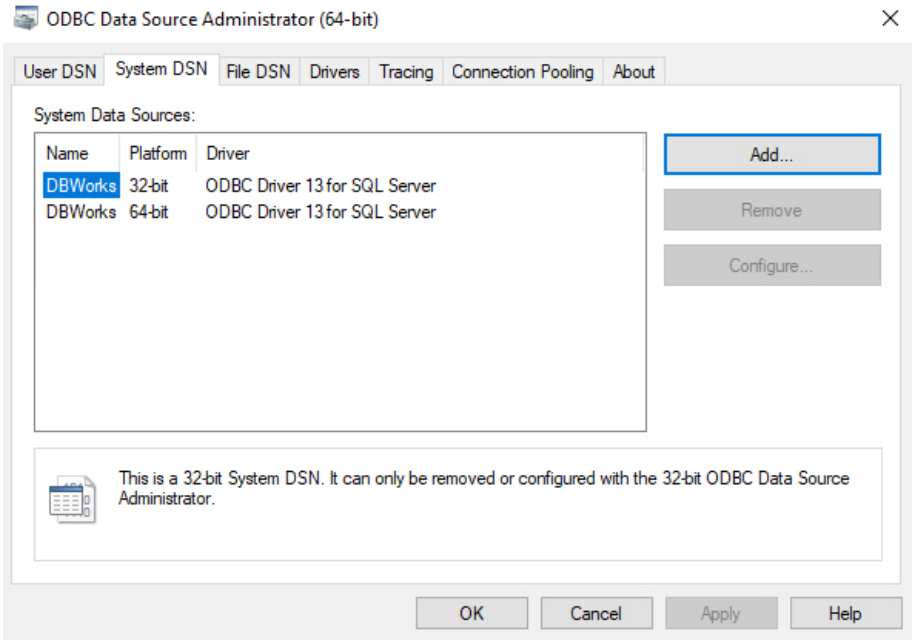


The **ODBC Data Source Administrator** dialog.

2. To ensure that the ODBC settings are correct (no matter who logs onto the workstation) change to the **System DSN** tab. Select the **[Add]** button to add the data source. Notice that the ODBC connection that was previously created for the 32-bit is displayed within the dialog. ***Caution: Do NOT create a data source within the System DSN tab with the same exact name as a data source within the User DSN tab.***



Continue to setup the ODBC connection for 64-bit by following the exact same steps as described earlier. Add the new connection, choose the **ODBC Driver 13 for SQL Server**, name the ODBC connection **DBWorks**, define the SQL Server, choose the **NT Authentication**, set the correct database, finish the setup and test the connection.



The ODBC connections have now been established.

ODBC Conclusion

The ODBC connection is critical for the functionality of MechWorks PDM (Integrated and Standalone). This connection must be established prior to running either application.

64 bit vs. 32 bit

It is critical to remember that if the workstation running MechWorks PDM (Integrated and Standalone) is using a 64 bit operating system, there are two locations to establish the ODBC settings.

1. Because MechWorks PDM utilizes some 32 bit functionality, establish the 32 bit ODBC settings for **DBWORKS** using the utility located in the **Control Panel -> System and Security -> Administrative Tools**.
2. In addition, the 64 bit ODBC connection for **DBWORKS** is established using the utility located in the **Control Panel -> System and Security -> Administrative Tools**.

MechWorks PDM Registration Codes

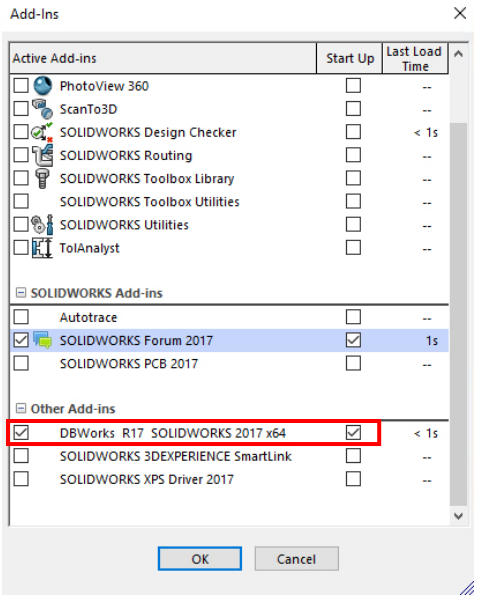
With MechWorks PDM installed, the **DBWORKS** database attached and the **ODBC** connections have been established it is now possible to begin using the application. Regardless of the version of MechWorks PDM that has been installed a **MechWorks PDM Registration Code** is required for properly licensing the application. If SolidWorks installed is installed, a **MechWorks PDM Registration Code** will be based on the CAD application's serial number. For all other CAD applications or for the Standalone Client the specific Client Registration wizard will need to be executed to produce the serial number that is derived from the workstation's hardware configuration. If the licenses are floating, then the MechWorks PDM License Manager must be installed and an appropriate MechWorks PDM Registration code will be provided.

Regardless of the version of MechWorks PDM or CAD application that has been installed, all installations of MechWorks PDM will run in a 30 day evaluation period. Before the 30 day evaluation period ends a proper MechWorks PDM Registration code must be entered to avoid having the application stop functioning.

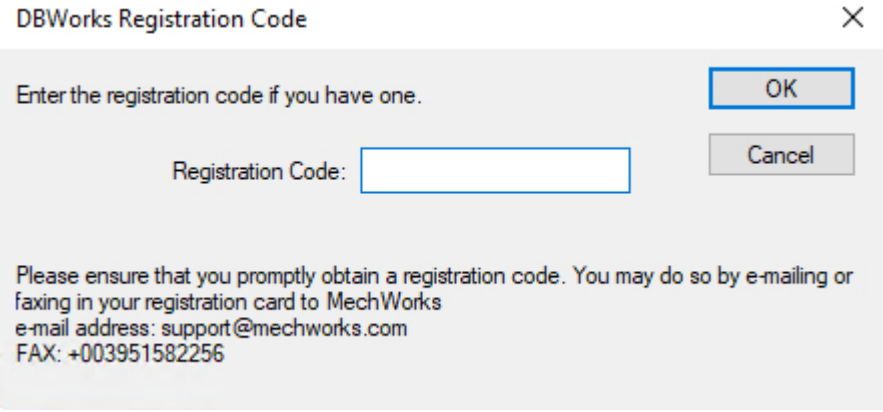
Note: For fixed seats of MechWorks PDM, the MechWorks PDM Registration Code is generated against a valid SolidWorks Serial Number. DBInventor, DBSolidEdge and Standalone Client require running the Client Registration Wizard to generate a serial number which will be used to generate a MechWorks PDM Registration Code. You will likely need Administration authority to enter the registration code.

DBWorks Integrated Registration code for fixed seats of SolidWorks

1. Start SolidWorks and go to the **Tools** drop-down menu and select its **Add-Ins** command. DBWorks is a SolidWorks **Gold Partner** application. To activate DBWorks, check the appropriate add-in selection boxes. There are two boxes that can be checked, one is the [Active Add-ins] box and the other is the [Start Up] box. Checking both boxes will ensure that DBWorks will start whenever SolidWorks is started. Select the **[OK]** button.

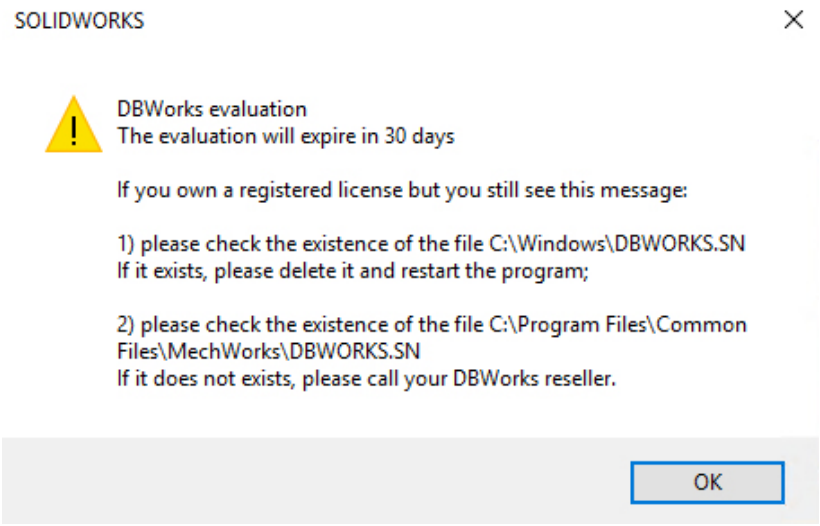


2. The DBWorks Registration Code dialog will now be displayed. If you have the assigned DBWorks Registration Code, enter it now. Without the Registration Code, the application will run as an **evaluation** version for 30 days. You will be required to enter a valid registration code prior to the expiration of the first 30 days, or the software will not continue to start.

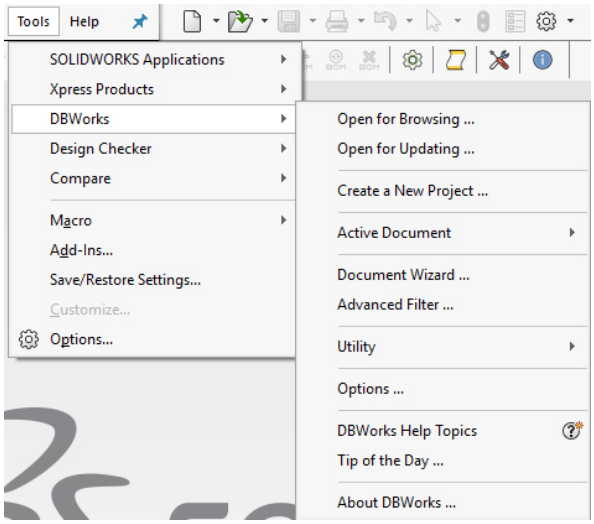


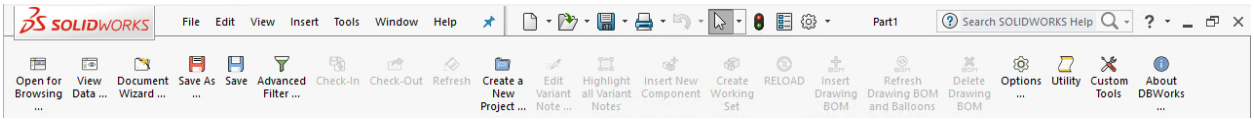
Note: The DBWorks Registration Code is generated against a valid SolidWorks Serial Number. You will likely need Administration authority to enter the registration code.

3. During an evaluation period, the system will prompt the user for a valid Registration Code each time DBWorks is activated. If no Registration Code is entered, a second prompt reminds the user how many days they have until the evaluation period expires.



4. Once activated, the DBWorks add-in will change two areas within SolidWorks. First, there is a DBWorks menu under the SolidWorks Tools pull down menu. Second, a DBWorks Command Manager has likely been added (this will not be displayed until a component is opened in SolidWorks).

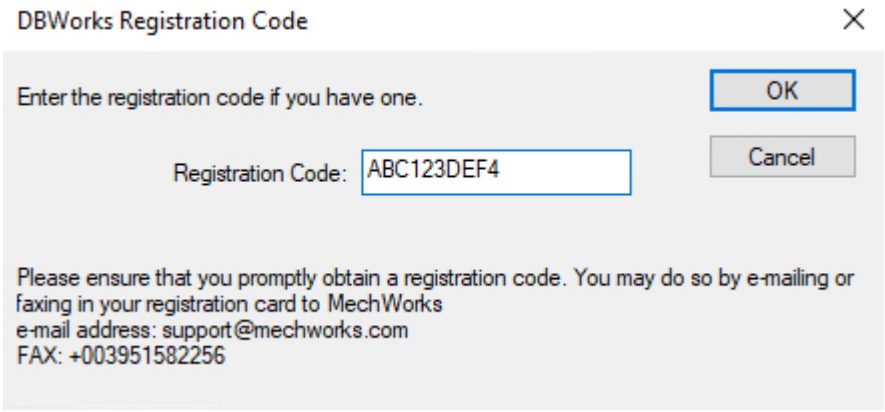


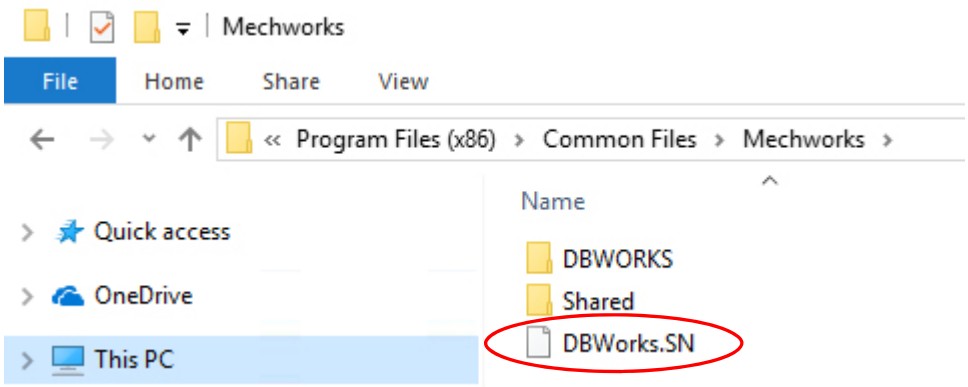


The DBWorks Command Manager is optional but recommended. The pull down menu is mandatory.

You can also turn on the DBWorks toolbar (the older toolbar versions).

Once a MechWorks PDM Registration code has been provided, start SolidWorks/MechWorks PDM. When prompted by the system, enter the ten digit Registration Code that has been provided for that exact SolidWorks serial number. NOTE: Enter it using **UPPER** case characters.



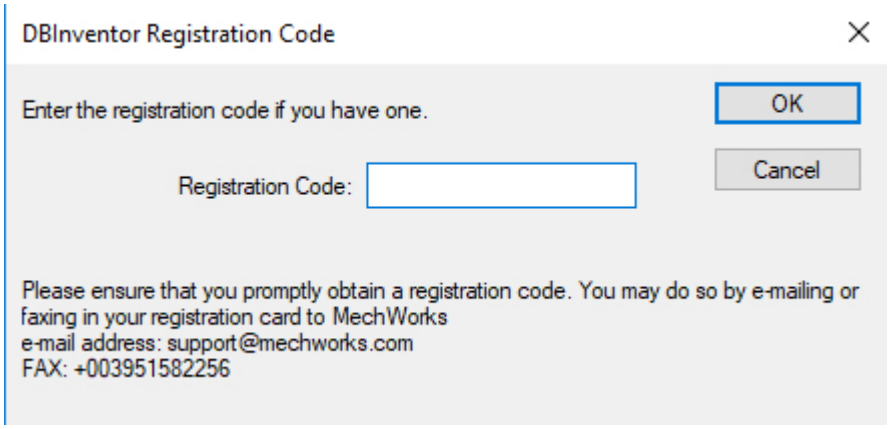


Entering the correct Registration Code will create a **MECHWORKS PDM.SN** file located within the **C:\Program Files (x86)\Common Files\MechWorks** folder. This file contains the Registration Code that was entered and must reside in this location for the license to be read.

DBInventor Integrated Registration Code for Inventor

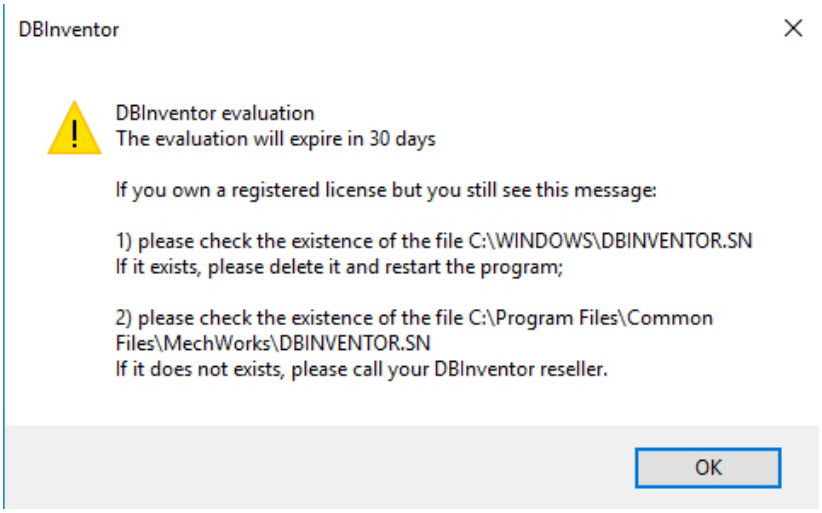
With the **ODBC** (Open Database Connectivity) data sources established, it is now possible to test DBInventor **Integrated**.

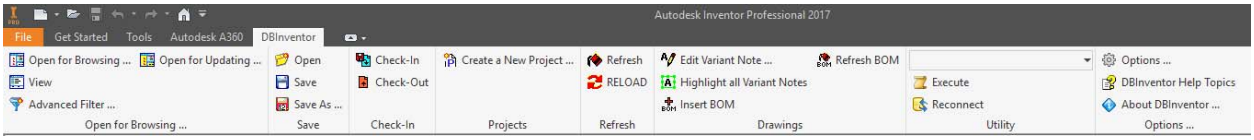
1. Start Inventor , by default DBInventor is already setup as an Add-In. The DBInventor Registration Code dialog will now be displayed. If you have the assigned DBInventor Registration Code, enter it now. Without the Registration Code, the application will run as an **evaluation** version for 30 days. You will be required to enter a valid registration code prior to the expiration of the first 30 days, or the software will not continue to start.



Note: The DBInventor Registration Code is generated by running the DBInventor registration wizard and submitting the results to your MechWorks PDM reseller.

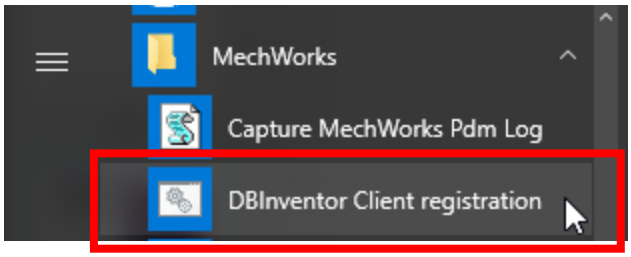
2. During an evaluation period, the system will prompt the user for a valid Registration Code each time DBInventor is activated. If no Registration Code is entered, a second prompt reminds the user how many days they have until the evaluation period expires.



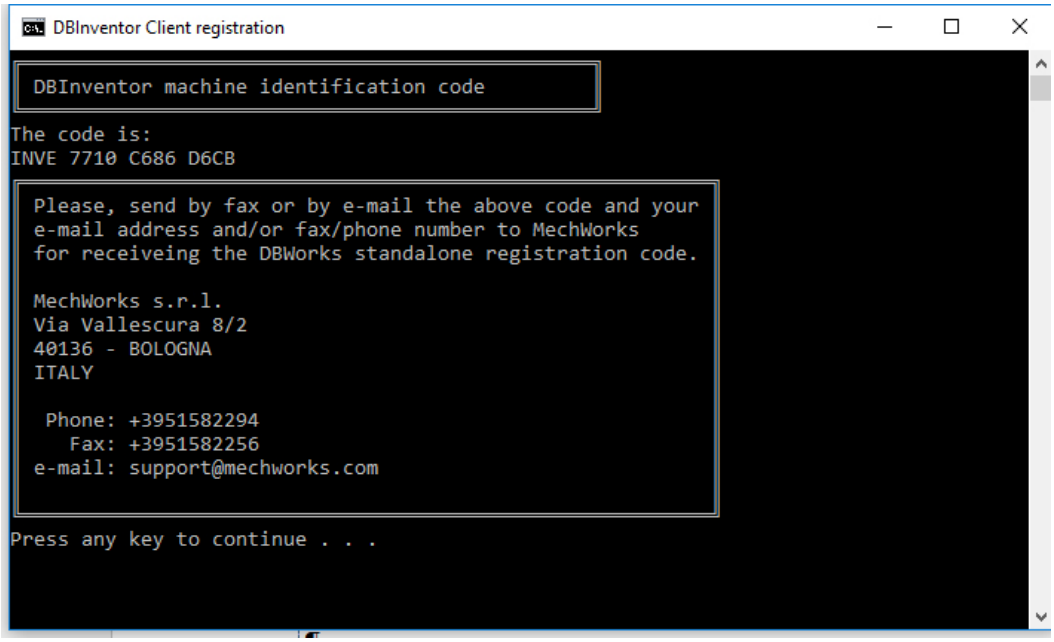


Once Inventor starts, the DBInventor add-in will add a DBInventor Ribbon.

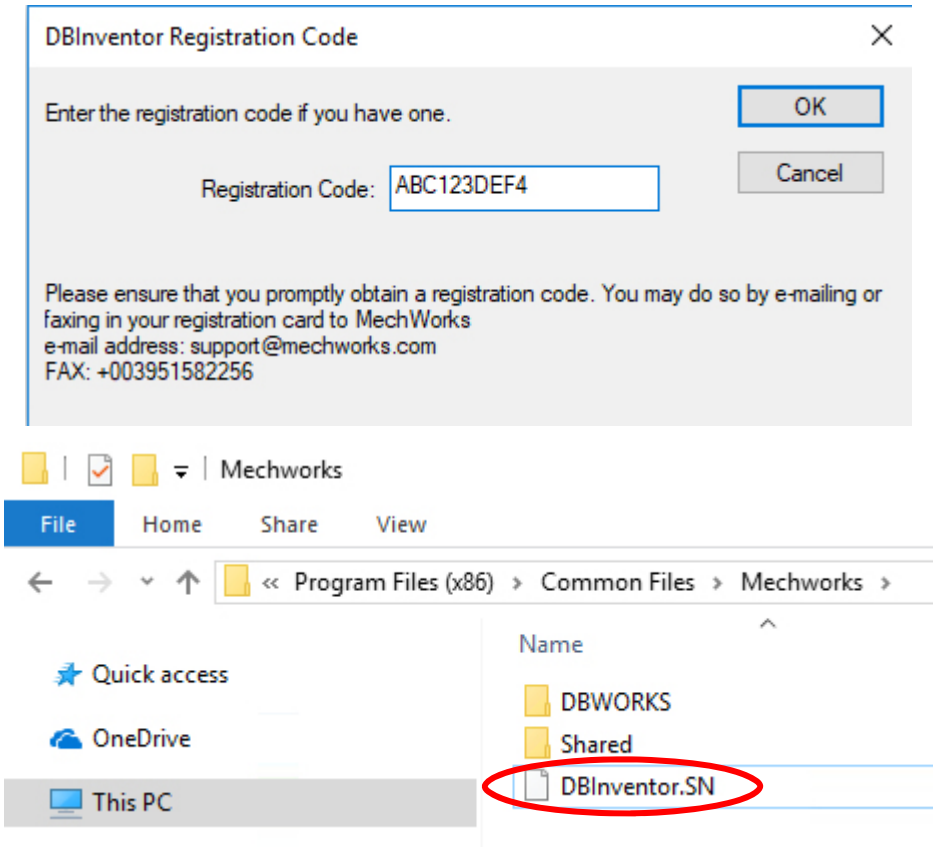
3. To generate a DBInventor code that is required for generating the DBInventor Registration Code run the **DBInventor registration** wizard by selecting the Windows Start button and locate the **MechWorks Apps**. Within this folder is the [**DBInventor Client registration]** shortcut.



4. The **DBInventor registration** wizard will produce a code specific for this machine. Take a screen shot of the results and submit this to your MechWorks PDM Reseller.



Once a DBInventor Registration code has been provided, start Inventor/DBInventor. When prompted by the system, enter the ten digit Registration Code that has been provided for that exact Inventor Registration code. NOTE: Enter it using **UPPER** case characters.

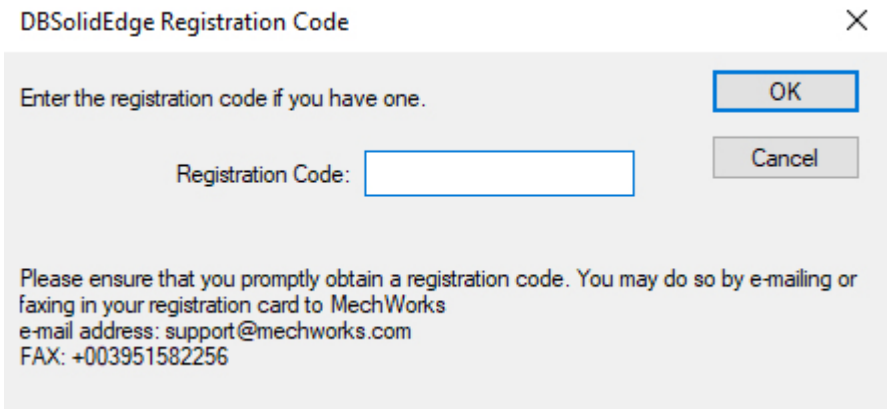


Entering the correct Registration Code will create a **DBInventor.SN** file located within the **C:\Program Files (x86)\Common Files\MechWorks** folder. This file contains the Registration Code that was entered and must reside in this location for the license to be read.

DBSolidEdge Integrated Registration Code for SolidEdge

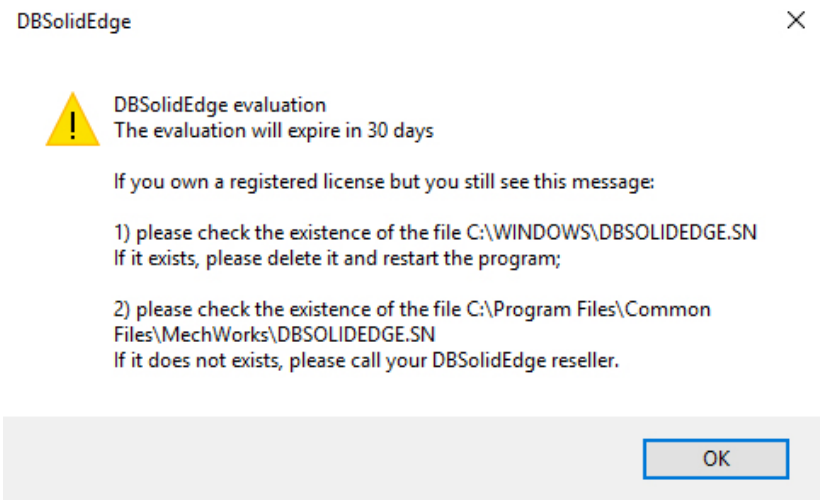
With the **ODBC** (Open Database Connectivity) data sources established, it is now possible to test DBSolidEdge **Integrated**.

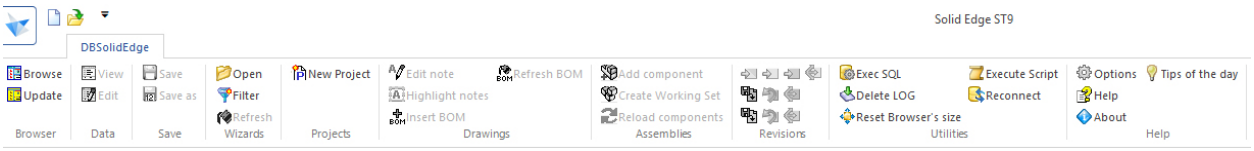
1. Start Solid Edge , by default DBSolidEdge is already setup as an Add-In. The DBSolidEdge Registration Code dialog will now be displayed. If you have the assigned DBSolidEdge Registration Code, enter it now. Without the Registration Code, the application will run as an **evaluation** version for 30 days. You will be required to enter a valid registration code prior to the expiration of the first 30 days, or the software will not continue to start.



Note: The DBSolidEdge Client Registration Code is generated by running the DBSolidEdge Client registration wizard and submitting the results to your MechWorks PDM reseller.

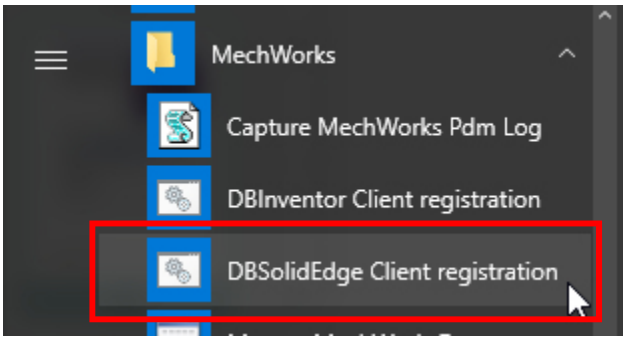
2. During an evaluation period, the system will prompt the user for a valid Registration Code each time DBSolidEdge is activated. If no Registration Code is entered, a second prompt reminds the user how many days they have until the evaluation period expires.



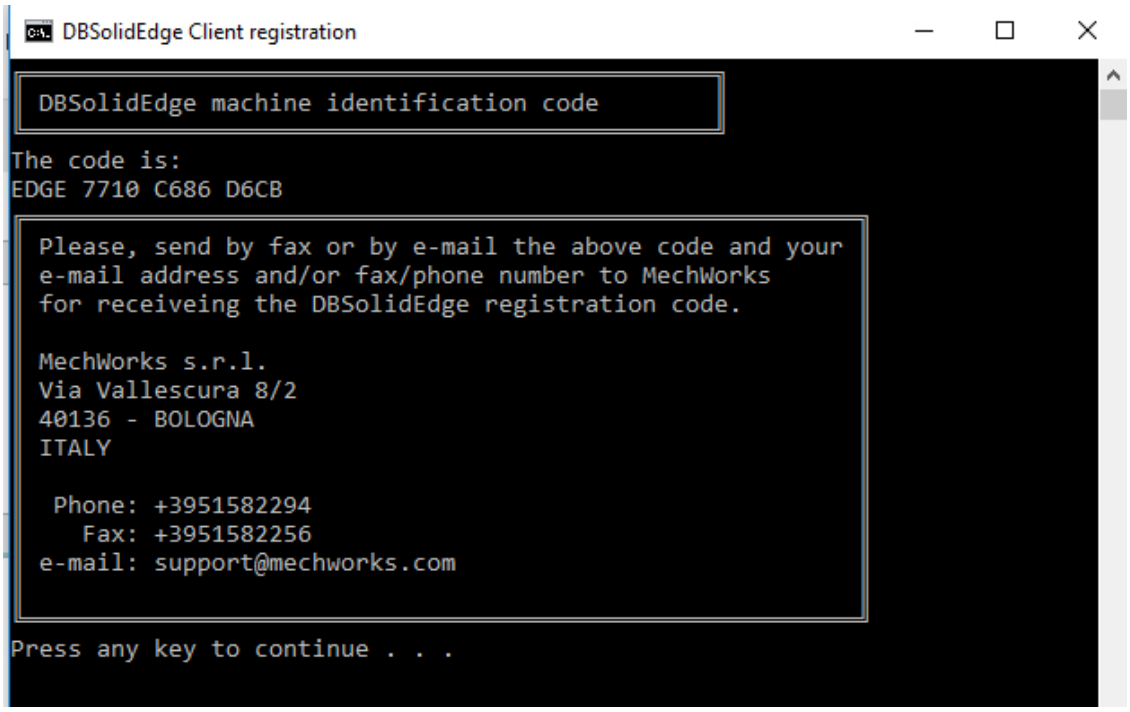


Once Solid Edge starts, the DBSolidEdge add-in will add the DBSolidEdge Quick Access toolbar

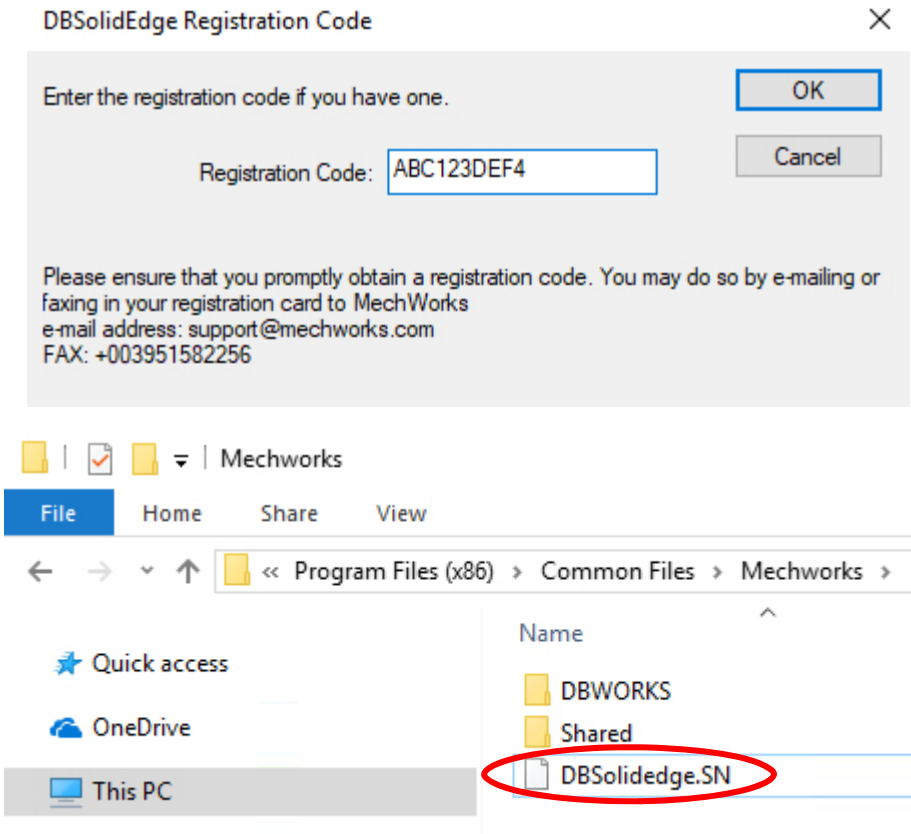
3. To generate a DBSolidEdge code that is required for generating the DBSolidEdge Client registration Code run the **DBSolidEdge Client registration** wizard by selecting the Windows Start button and locate the **MechWorks Apps**. Within this folder is the [**DBSolidEdge Client registration]** shortcut



4. The **DBSolidEdge Client registration** wizard will produce a code specific for this machine. Take a screen shot of the results and submit this to your MechWorks PDM Reseller.



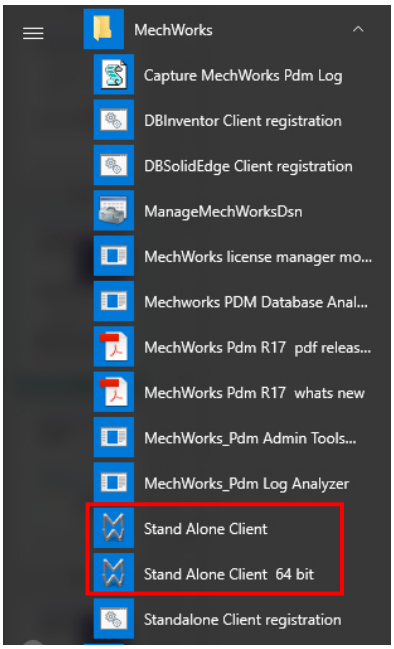
Once a DBSolidEdge Registration code has been provided, start Solid Edge/DBSolidEdge. When prompted by the system, enter the ten digit Registration Code that has been provided for that exact DBSolidEdge Registration code. NOTE: Enter it using **UPPER** case characters.



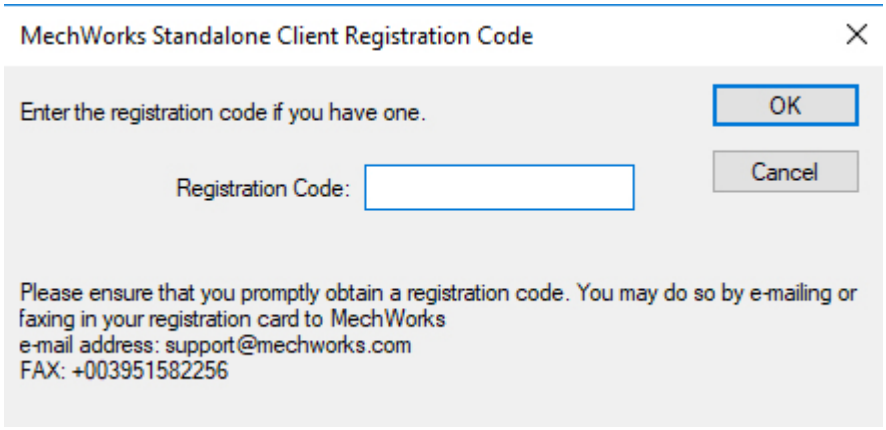
Entering the correct Registration Code will create a **DBSolidEdge.SN** file located within the **C:\Program Files (x86)\Common Files\MechWorks** folder. This file contains the Registration Code that was entered and must reside in this location for the license to be read.

MechWorks PDM Standalone Client Registration Code

1. Start the Standalone Client by selecting the Windows Start button and locate the **MechWorks Apps**. Within this folder is the **[Standalone Client]** shortcut. Select the shortcut to start the Standalone Client.

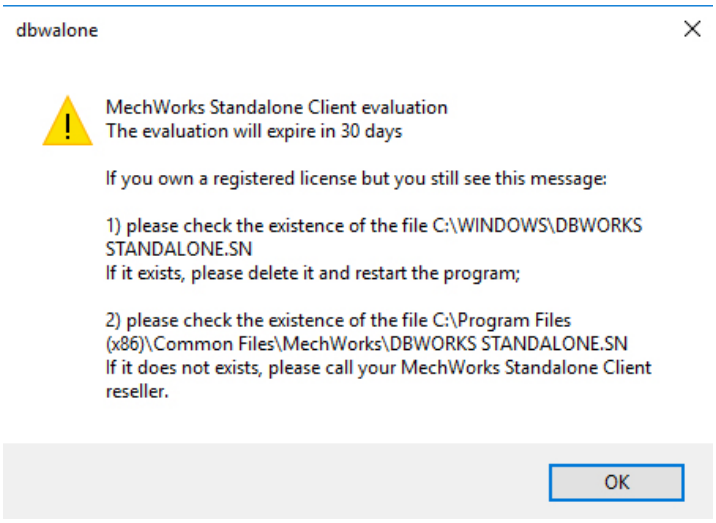


2. If the Standalone Client is the only version of MechWorks PDM installed on the workstation, the Standalone Registration Code dialog will now be displayed. If you have the assigned Standalone Registration Code, enter it now. Without the Registration Code, the application will run as an **evaluation** version for 30 days. You will be required to enter a valid registration code prior to the expiration of the first 30 days, or the software will not continue to start.

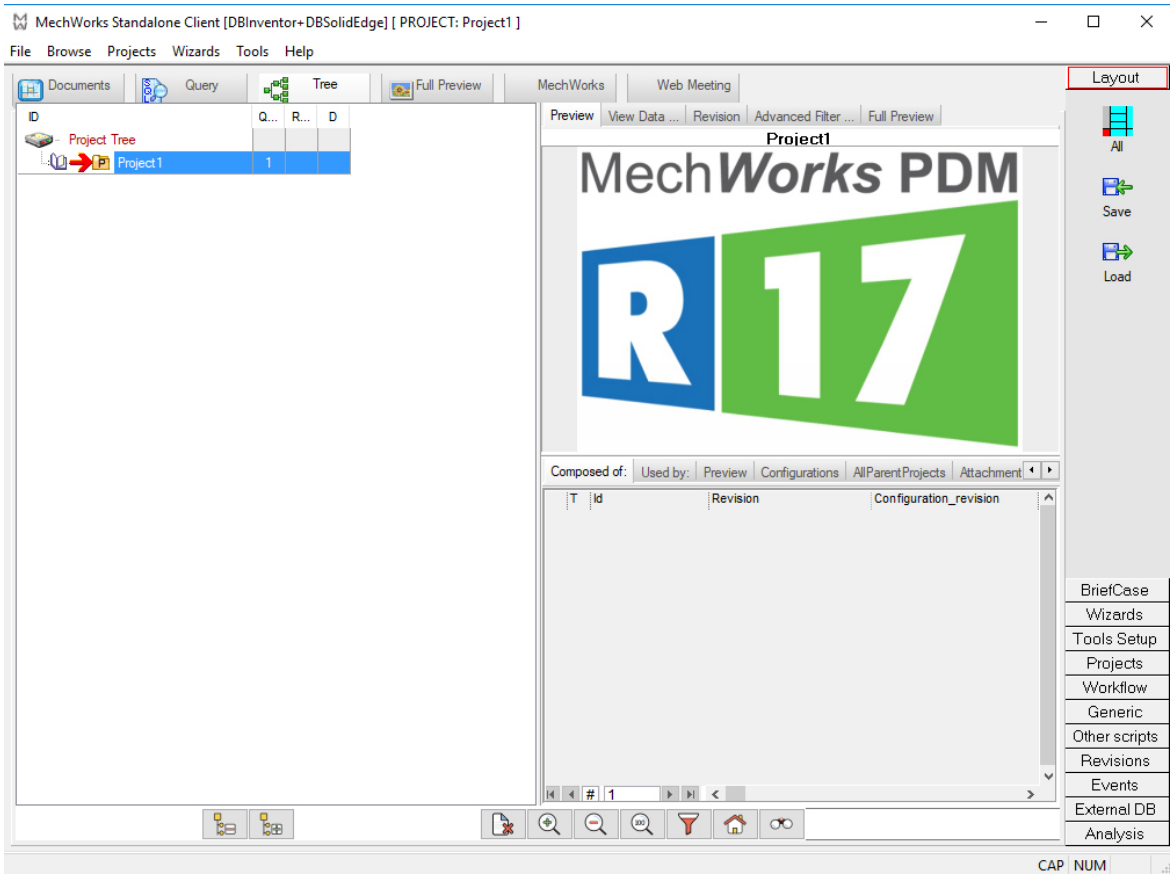


Note: If this application of Standalone was installed on the same workstation as DBWorks\DBInventor\DBSolidEdge Integrated, enter the same Registration Code used for DBWorks\DBInventor\DBSolidEdge Integrated for the Standalone Client. If Standalone is the only MechWorks PDM application installed, a unique Registration Code will need to be generated by first running the Standalone Client registration utility and submitting its results to your MechWorks PDM reseller.

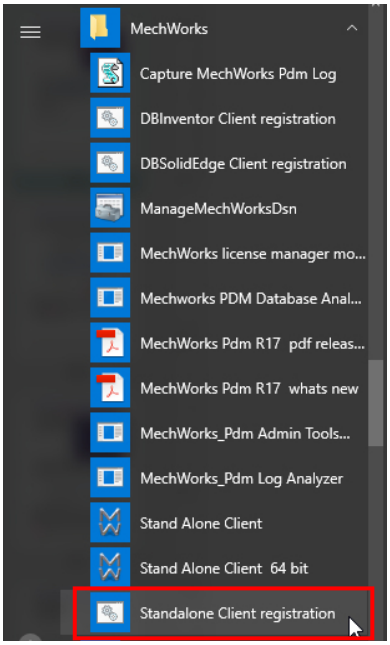
3. During an evaluation period, the system will prompt the user for a valid Registration Code each time Standalone Client is activated. If no Registration Code is entered, a second prompt reminds the user how many days they have until the evaluation period expires.



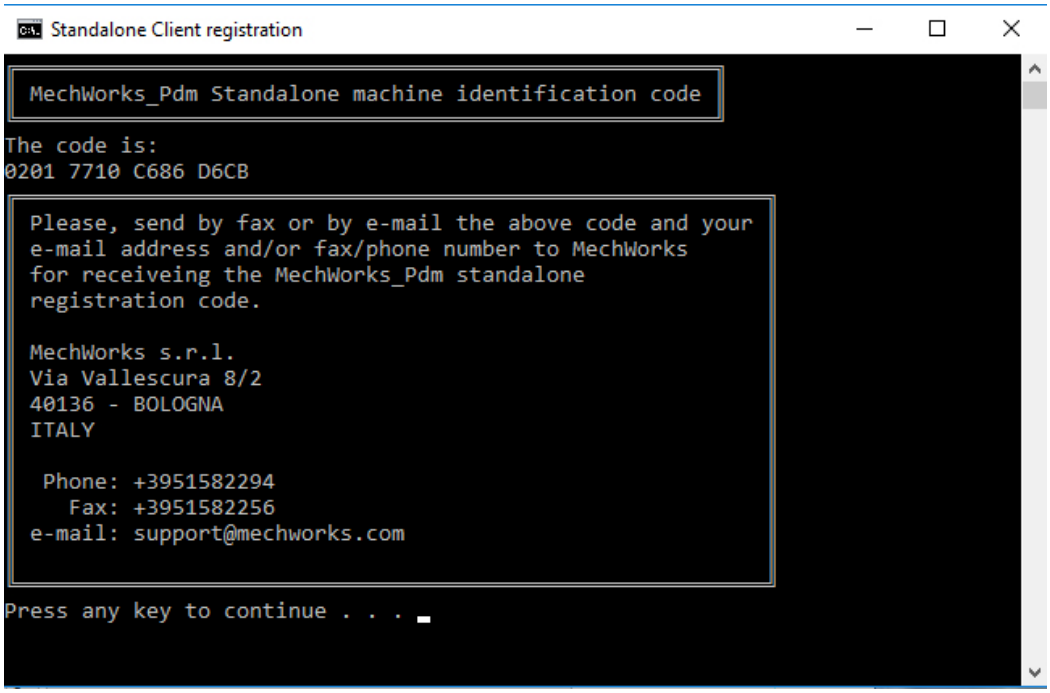
4. The Standalone Client will now open, connected to the same database as the Integrated version of MechWorks PDM.



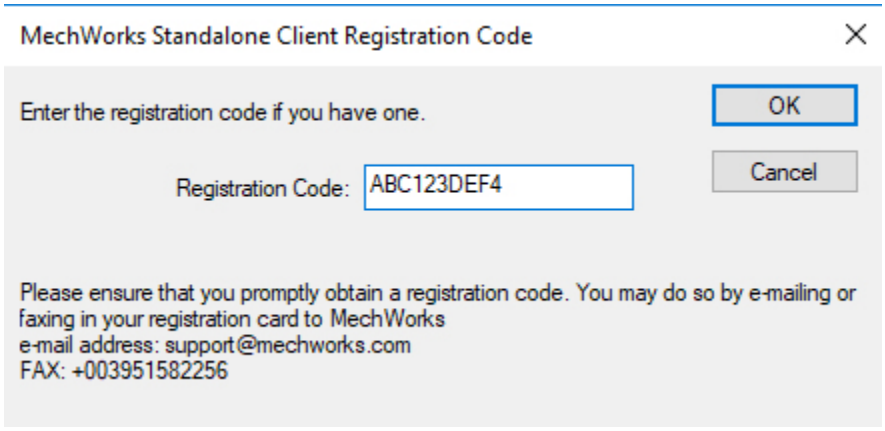
5. To generate a Standalone Client registration code run the **Standalone Client registration** wizard by selecting the Windows Start button and locate the **MechWorks** Apps. Within this folder is the [**Standalone Client registration]** shortcut.

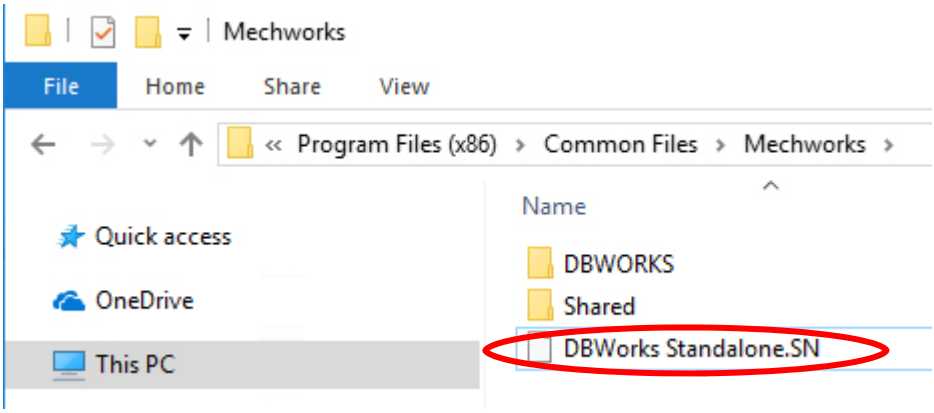


6. The **Standalone Client registration** wizard will produce a code specific for this machine. Take a screen shot of the results and submit this to your MechWorks PDM Reseller.



Once a Standalone Client Registration code has been provided, start the Standalone Client. When prompted by the system, enter the ten digit Registration Code that has been provided. NOTE: Enter it using **UPPER** case characters.



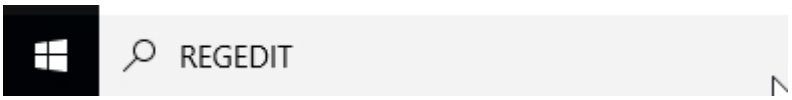


Entering the correct Registration Code will create a **MechWorks PDM Standalone.SN** file located within the **C:\Program Files (x86)\Common Files\MechWorks** folder. This file contains the Registration Code that was entered and must reside in this location for the license to be read.

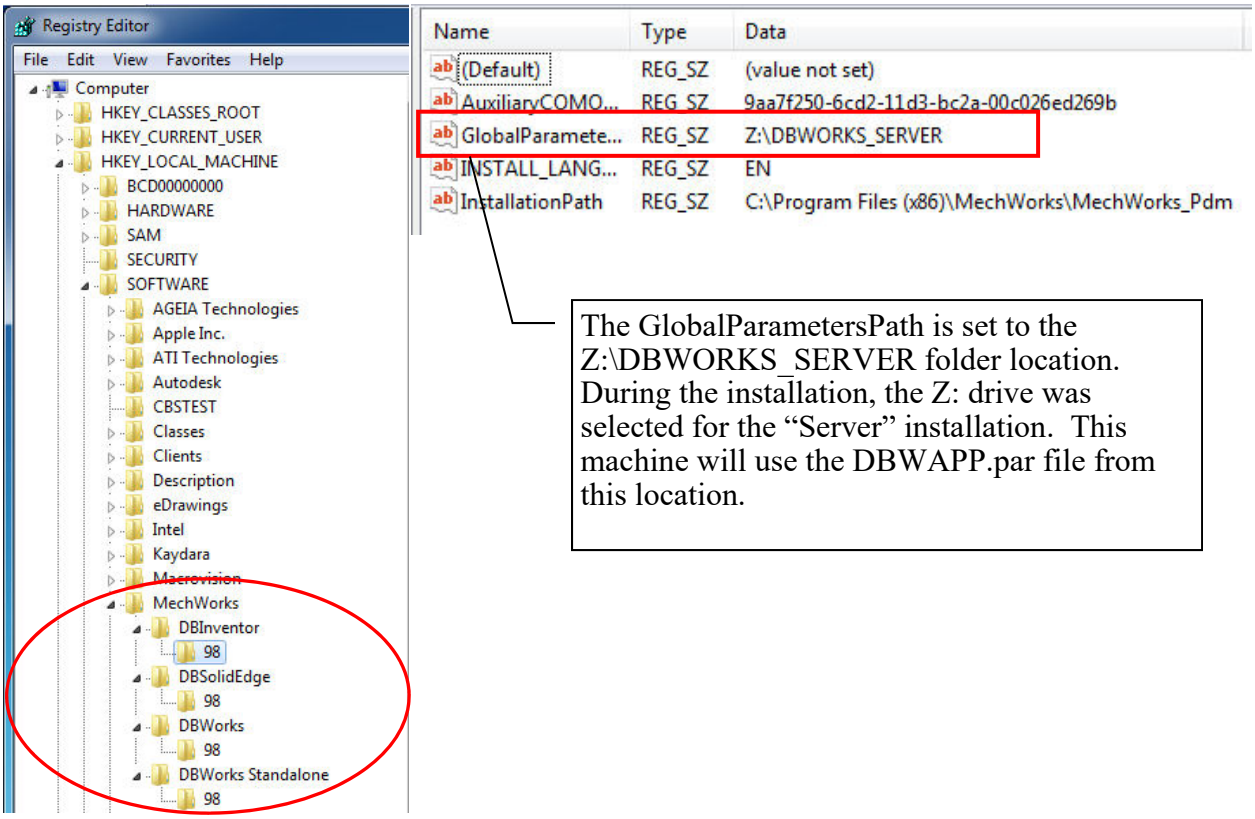
Registry Settings

Every installation of MechWorks PDM (Integrated and Standalone) references a parameter file location for its ***DBWAPP.PAR*** file (this file is explained in a later lesson). To validate that the application is directed to the correct location, the Windows Registry setting for the **GlobalParametersPath** must be configured properly, to use the mapped drive location that was selected during the Client installation process.

1. To determine if the **GlobalParametersPath** is set correctly, type **regedit** in the Windows Run command line. **Caution: When editing the Registry, it is extremely important that you are very careful. Editing registry settings incorrectly can cause fatal workstation problems.**



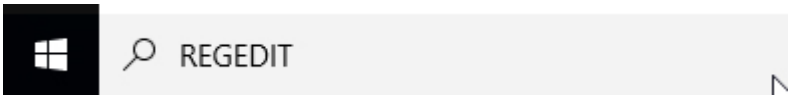
2. Browse to the **HKEY_LOCAL_MACHINE ⇨ SOFTWARE ⇨ MechWorks** registry key. Each version of MechWorks PDM has it own registry key. Within this key, there is a sub-key **98**. The **GlobalParametersPath** is defined and its value is where the path is configured for determining where the selected application will read its critical **DBWAPP.PAR** file.



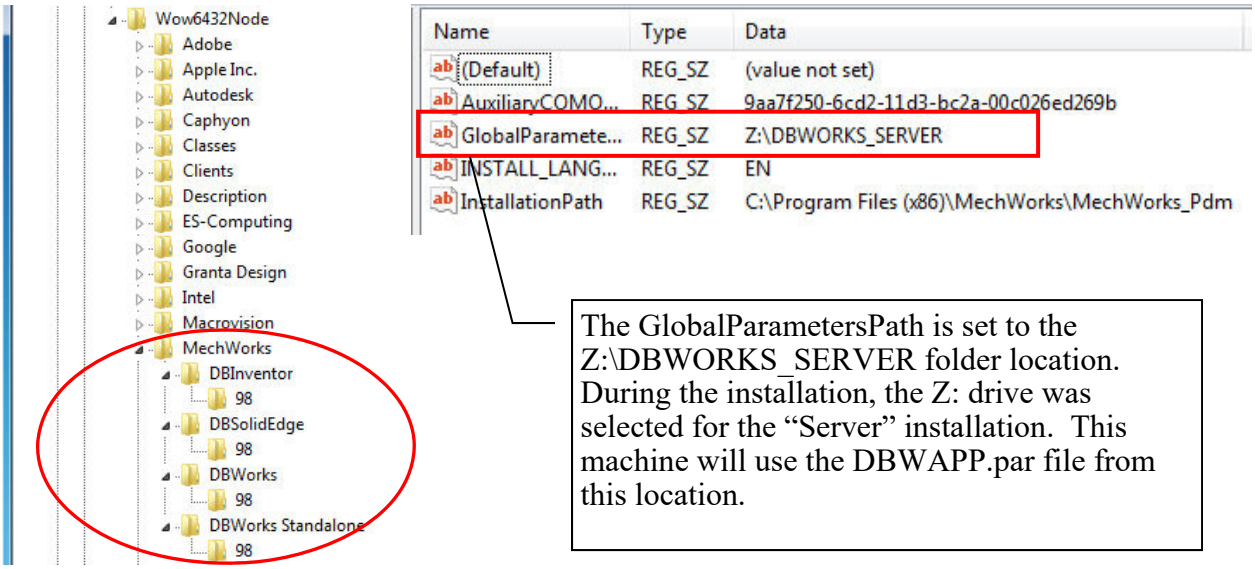
Registry Settings x64

Every installation of MechWorks PDM (Integrated and Standalone) references a parameter file location for its ***DBWAPP.PAR*** file (this file is explained in a later lesson). To validate that the application is directed to the correct location, the Windows Registry setting for the **GlobalParametersPath** must be configured properly, to use the mapped drive location that was selected during the Client installation process.

1. To determine if the **GlobalParametersPath** is set correctly, type **regedit** in the Windows Run command line. **Caution: When editing the Registry, it is extremely important that you are very careful. Editing registry settings incorrectly can cause fatal workstation problems.**



2. Browse to the **HKEY_LOCAL_MACHINE ⇨ SOFTWARE ⇨ Wow6432Node ⇨ MechWorks** registry key. Each version of MechWorks PDM has it own registry key. Within this key, there is a sub-key **98**. The **GlobalParametersPath** is defined and its value is where the path is configured for determining where the selected application will read its critical **DBWAPP.PAR** file.

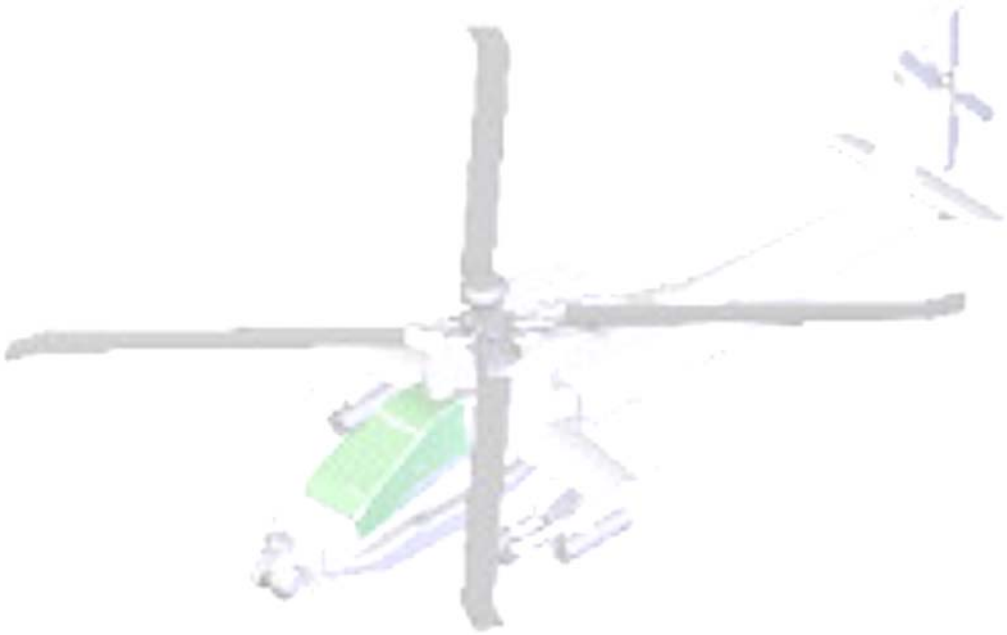



Intentionally Blank
as a separator

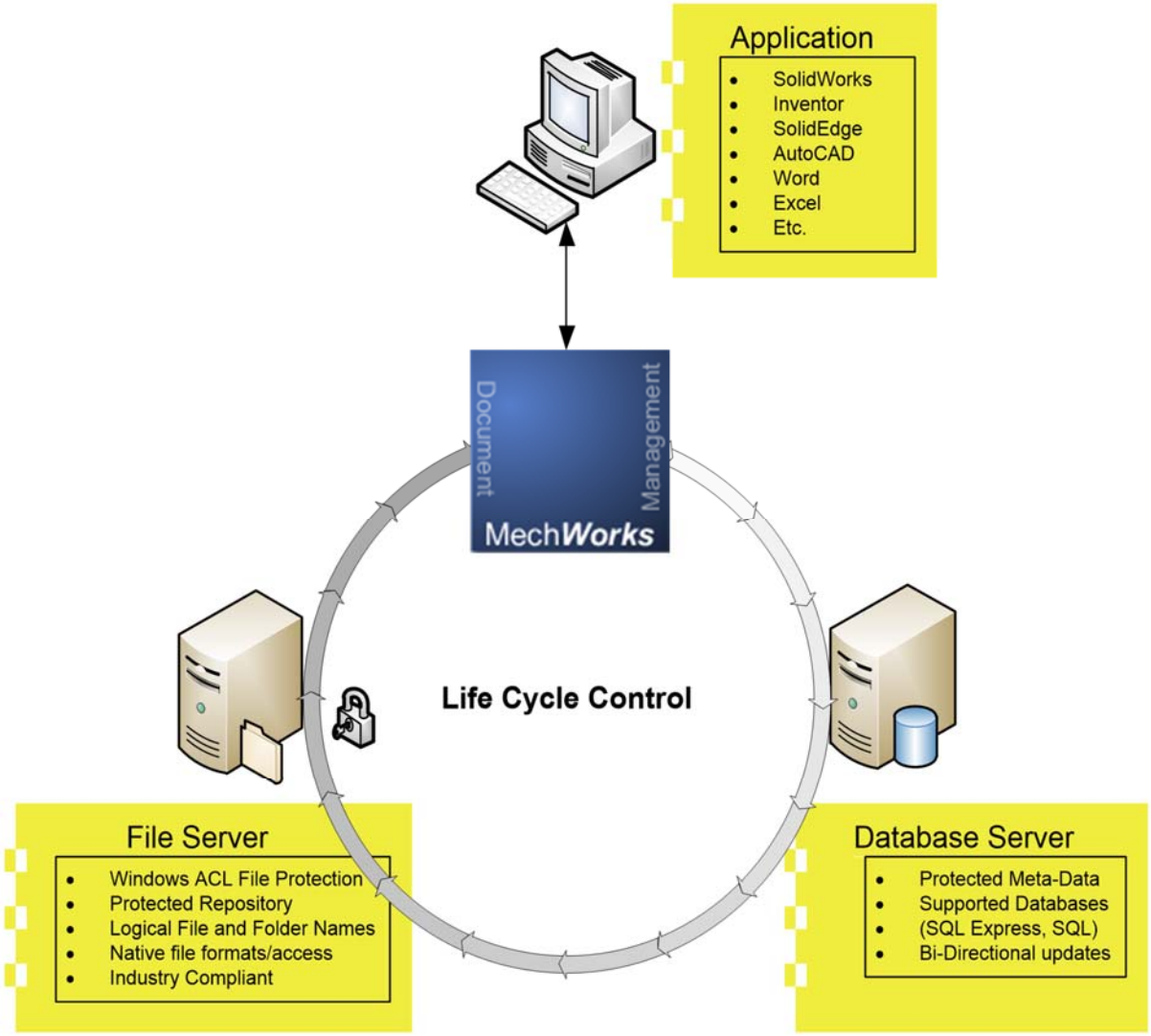

Intentionally Blank
as a separator

Chapter 3

Understanding MechWorks PDM



ROLE



MechWorks PDM: (The common thread.)

You can see from the figure above that MechWorks PDM is the common thread. The role of MechWorks PDM is to act as the communication layer between the other three primary participants (Application, File/Object, and the Database). MechWorks PDM functions as a dispatching agent and **does not** attempt to take over tasks that the other three applications were developed to handle (fully integrated). MechWorks PDM is built on a suite of industry standard tools to ensure the tightest possible integration between the Database, Operating System, and CAD - ***bonding them together as a single unit for Design Integrity***.

Each time you save a CAD document, MechWorks PDM collects the available data about the current CAD document. This information may include physical properties, system information such as date, time, user etc… and hierarchical relations which are vital to drawings, assemblies and sub-assemblies. In the previous illustration, you will see that this information is entered in the database and can be displayed in a number of locations from the MechWorks PDM interface to a BOM to the file properties. A journal of events is also created to trace all the properties. Most important, you are given a chance to add information about custom attributes that are valuable to you, but are not necessarily exposed directly by the CAD (material, cost, etc…).

All of the information gathered is stored into the database to re-create the same hierarchical relations of the type 'contains' that you naturally established by inserting one part or assembly into another assembly or drawing. This process does not require any intervention by the user as MechWorks PDM can collect all the necessary information directly from the CAD in a transparent way. This approach to data gathering allows you to store data in a transparent mode at save time and to be able to browse the database as a hierarchical structure afterwards.

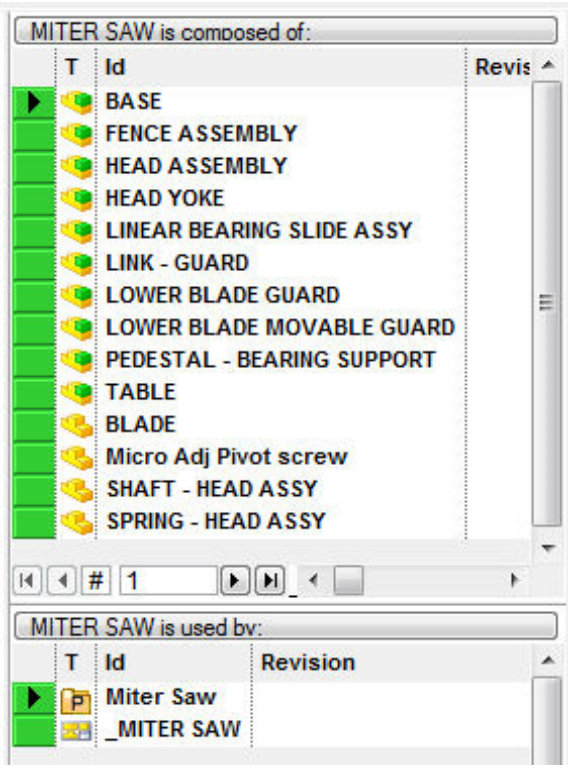
MechWorks PDM is a non-intrusive tool that handles all I/O interfacing while exposing itself as the main control panel for all of your records.

- **How does MechWorks PDM handle CAD relations?**

Hierarchical relations

Each time you save a document containing other sub-documents (i.e. assembly containing sub-assemblies and parts, a drawing containing parts, etc…) MechWorks PDM maintains these sub-documents relationships in its database.

Because of the way the CAD itself has been designed, the same part can be included in several documents. All these 'contains' relations are stored into the database, allowing you to browse it efficiently and quickly in a hierarchical way, starting from a project and going down to each document (CAD documents or Generic documents) or from a part up through assemblies, drawings or projects. Moreover, you can freely move documents to other directories or computers without breaking any existing link.



- **How does MechWorks PDM handle Configurations?**

Most PDM systems do not handle configurations. MechWorks PDM allows you to maximize your CAD documents with the ability to use the CAD configurations.

Configurations and support for revisions of configured documents
It is possible to create revisions of documents with configurations written as distinct records.

The basics
The revision's support applies to the file where the document is stored. Therefore, all the records pointing to the same file share the same revision state and revision value, but each configuration can have a unique configuration-revision value. This implies that when a document is checked-out, all of the other records pointing to other configurations of the same document are checked-out just as well. Consistent with other operations related to revisions. As noted, MechWorks PDM does allow for independent revisions for each configuration as well.

How to enable this support
You must enable the following options:

Save configurations as distinct records

How to add a new configuration to a released document
To add a new configuration to a released document with many existing configurations, you need to checkout the document (this means that all configured records will be checked-out) with the document open, add the new configuration in the CAD configuration tab, make it the current configuration. To make it the current configuration, double-click on the newly created configuration, do any feature modifications that you require, and finally, Save and CHECK-IN the new configuration. A new record is created for the new configuration with the same revision number as the other ones. When you check-in again, the entire record set related to the configurations of the same file will be checked-in.

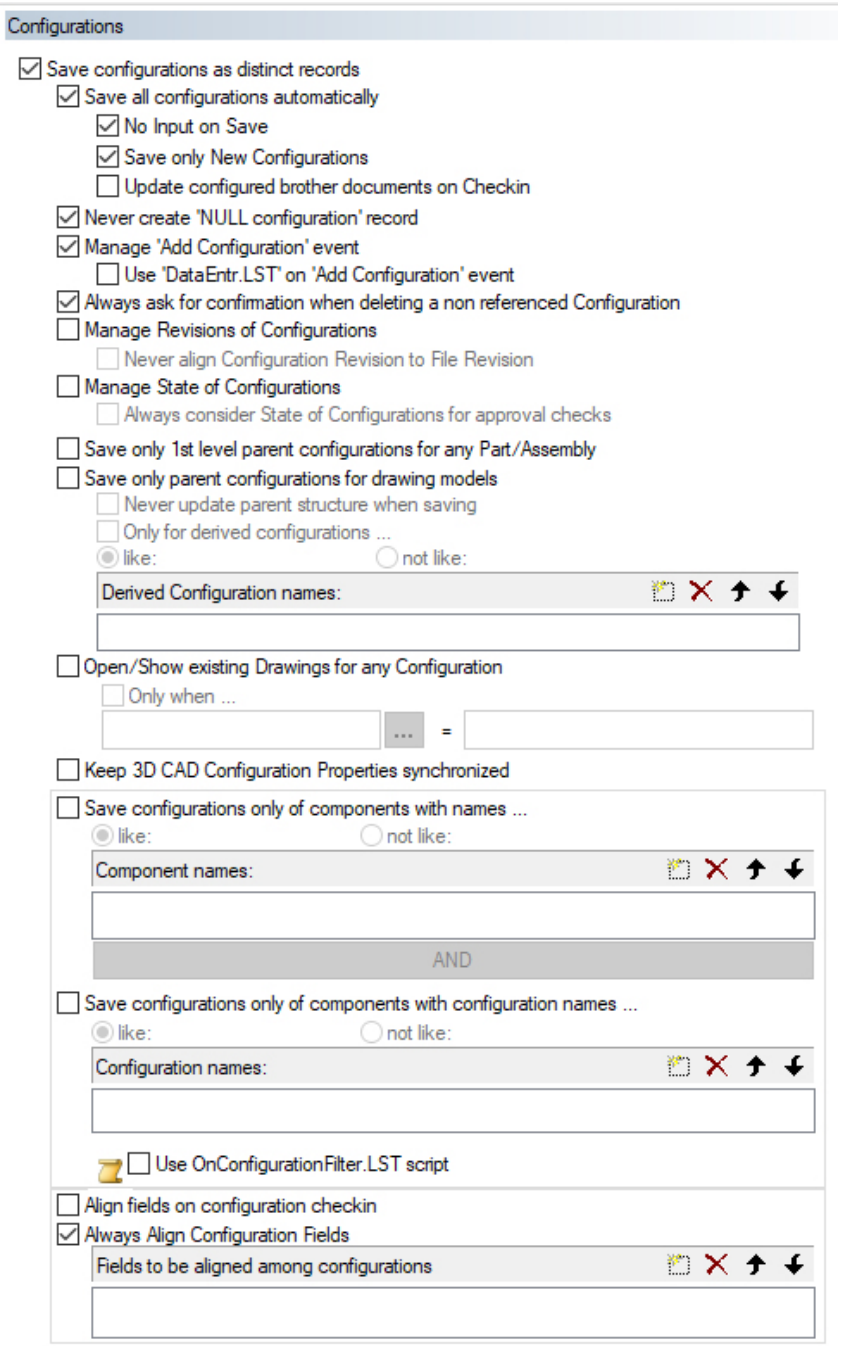
Should I save distinct configurations as distinct records?
If you do not need to apply the revision process to several configurations of a given document, you may choose to keep the ☐ *Save configurations as distinct records* option unchecked.

IMPORTANT: Once you have decided upon a method and you have used MechWorks PDM for some time, you will not be able to change your method for the existing documents, unless you delete them and redo the saving job and data input from scratch.

One last note on configurations.

Depending on your CAD version, it is possible to save configurations based on the configuration names or the filename. These options have been introduced to allow for separate management of files belonging to part libraries. This allows for mixed (yet still clearly defined) operational approaches.

For such documents, the correct handling of configurations as a distinct record is necessary, but you are not likely to have a part from a library undergo a revision process as all of these types of parts are certainly final. If it **is** necessary to add a revision to one specific configuration, MechWorks PDM allows for such instances.

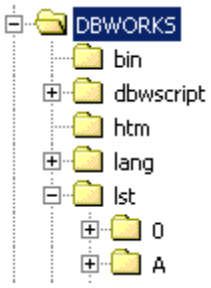


- **What is a Project to MechWorks PDM?**

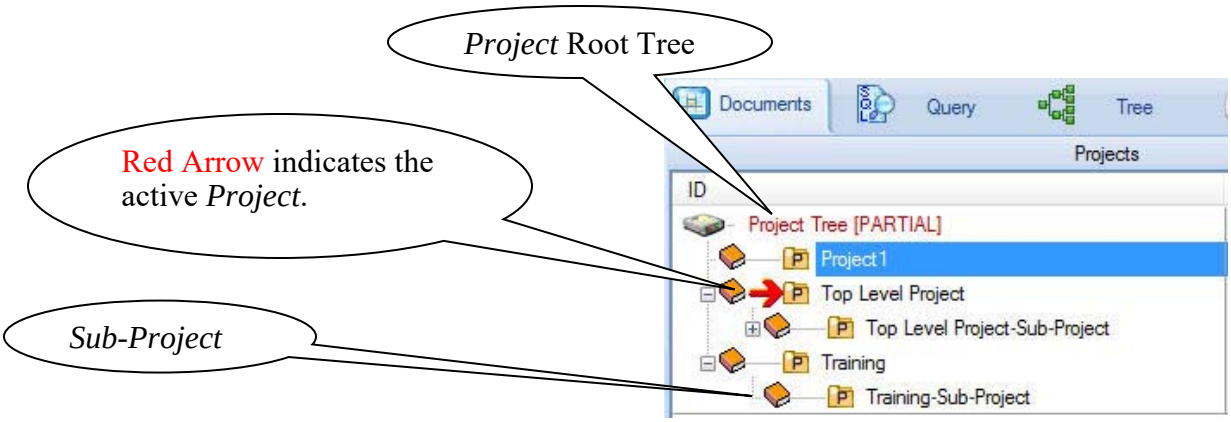
MechWorks PDM provides *Projects* to allow for a **logical collection** of documents. The logical collection of documents is defined by the user and can be a mixture of file types including SolidWorks' ("SLDASM", "SLDPRT", and "SLDDRW") as well as any other file types ("DOC", "XLS","TXT", "BMP", "PDF", etc.). *Projects* can contain *sub-projects,* providing a hierarchical container for a finite definition of collected documents. Additional *Project* clarifications:

- Only one *Project* is considered the sessions *active Project* at any given time.
- *Projects* can have user or group access rights applied to them.
- The "top" *Project* known as the "*Project Root Tree*" is not user definable.
- Documents can belong to more than one *Project*.
- *Sub-Projects* can belong to more than one *Project* (not a common practice).

ANALOGY: *Projects* are similar to Windows® system folders. Just as MechWorks PDM provides a single top level "*Project Root Tree*" Windows® also provides a single partition root, often seen as "C:\" or "D:\".

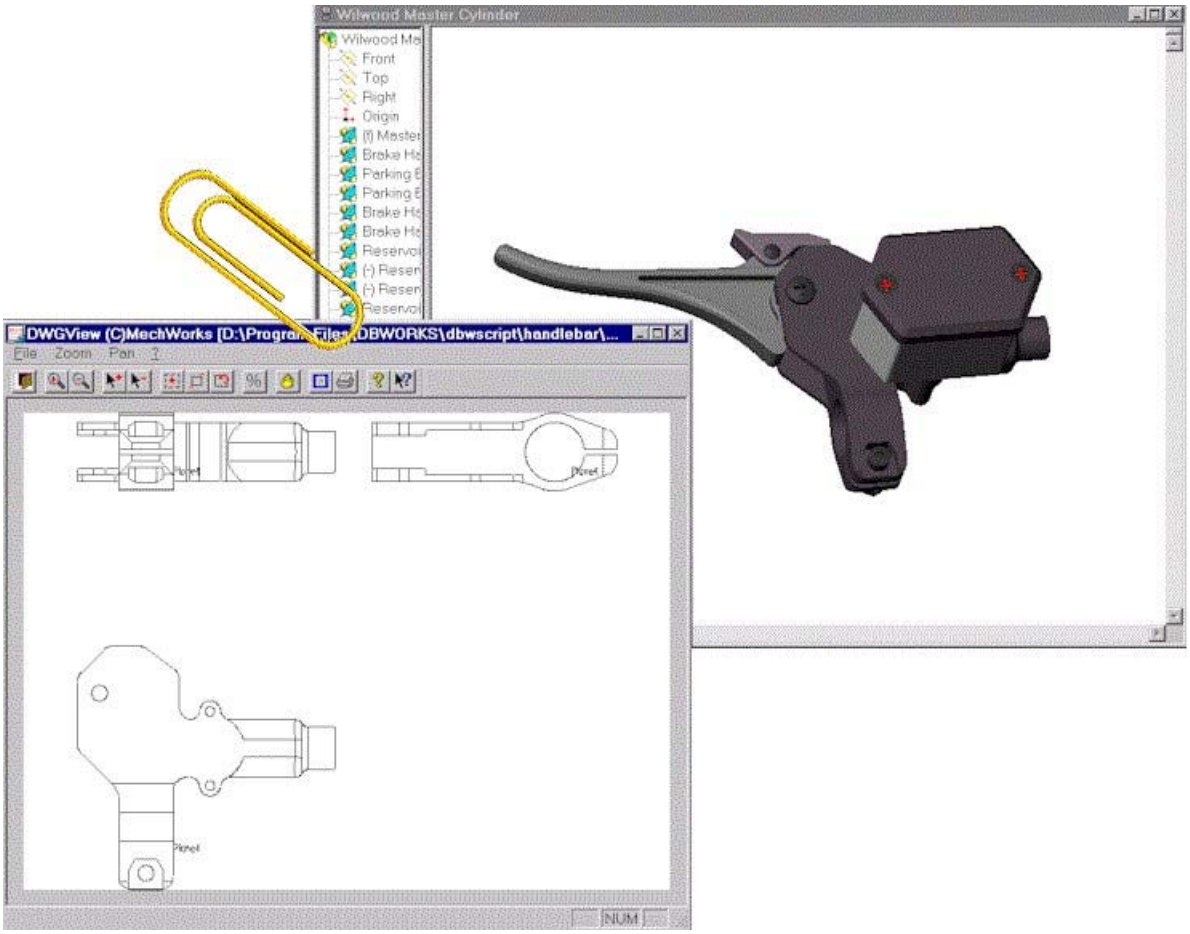


Project Tree



- **What about non-CAD documents?**

MechWorks PDM allow you to associate non-CAD documents to any project or CAD documents which are related to it. This feature enables you to integrate a part/assembly/ drawing or project with text notes, voice notes, Excel sheets, Word documents, bitmaps, AutoCAD files or any other file that should accompany the SolidWorks document for future reference or information sharing with a team.



Where possible, MechWorks PDM displays a small preview of the document in its preview window.

No matter the file type of the document, you can open it directly from MechWorks PDM by clicking **Open** in its pop-up menu. MechWorks PDM will run the application registered at the Operating System level and open it as if you double-clicked the file in Windows Explorer.

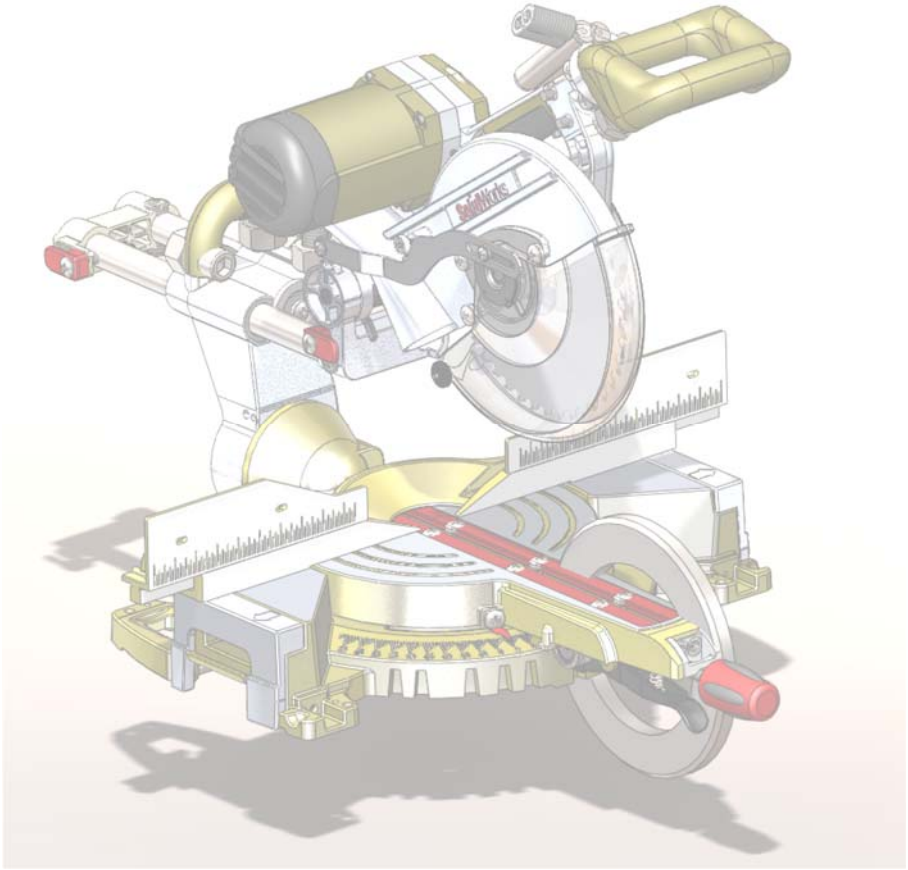
Special case files are AutoCAD files. For these files, MechWorks PDM provides an internal viewer which allows zooming and printing. For further information regarding this feature, please see the MechWorks PDM help topic: *The DWG files viewer*.


Intentionally Blank
as a separator


Intentionally Blank
as a separator

Chapter 4

User Interface



MechWorks PDM is a document management application that will allow for security of files. It is fully integrated with SolidWorks and includes the following benefits (and more):

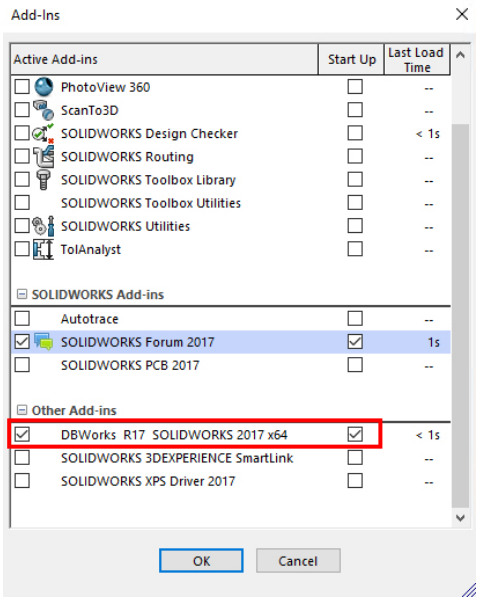
- MechWorks PDM maintains a database record for your CAD documents.
- MechWorks PDM maintains a history of changes made to its records.
- MechWorks PDM enforces revision control, along with "Check-In / Check-Out" functions.
- MechWorks PDM organizes documents by projects.
- MechWorks PDM has advanced search (query) tools.

Invoke MechWorks PDM

Once MechWorks PDM has been installed on your PC, you may need to activate MechWorks PDM. From SolidWorks, use the **Tools** *pull down* menu and choose the **Add-Ins…** command. From the **Add-Ins…** dialog box, *check* the box next to DBWorks R17 then select the **OK** command.

You will need to reverse this procedure if you need to work with SolidWorks, without using MechWorks PDM.

It is not necessary to activate MechWorks PDM in Inventor or Solid Edge; MechWorks PDM is automatically added to these CAD packages after installation.

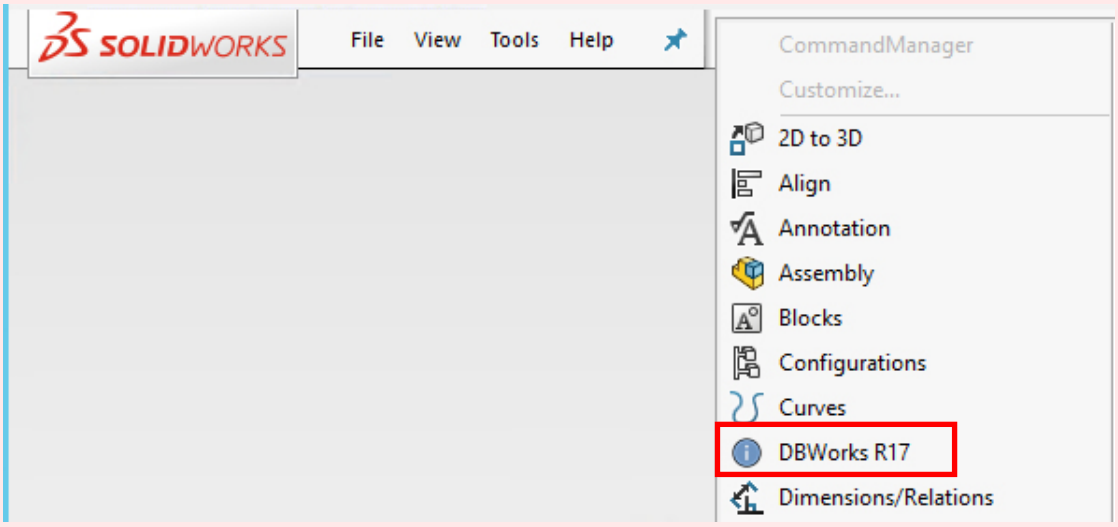


With MechWorks PDM activated, SolidWorks user's interface may display four areas of change:

- MechWorks PDM "Toolbar"
- MechWorks PDM "Command Manager"
- MechWorks PDM "Menu"
- MechWorks PDM "Feature Manager tab"

Toolbar

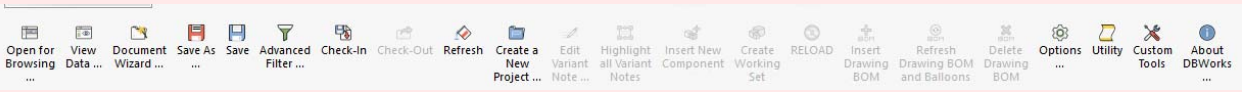
By default, the toolbar may not be displayed when DBWorks is first activated in the SolidWorks Add-Ins. Because the toolbar is compliant SolidWorks standards it must be activated and then can be repositioned by dragging and dropping it. You will also notice that some of its icons are grayed-out because the toolbar is context sensitive and only activates relative commands to the user's current session. As you hover your mouse over any of its icons, the command's name will be displayed. *NOTE*: Optionally, the toolbar does not need to be displayed, but can help with easier access to the DBWorks Browser.



If the DBWorks Toolbar is activated it can be placed anywhere on the screen just like any other SolidWorks Toolbar. It can also be modified (adding or removing tools) in the same manner that any other SolidWorks toolbar is modified.

MechWorks PDM Command Manager

The MechWorks PDM Command Manager tool bar is compliant to SolidWorks standards and can be repositioned by dragging and dropping it. You will also notice that some of its icons are grayed-out because the Command Manager is context sensitive and only activates relative commands to the user's current session. As you hover your mouse over any of its icons, the command's name will be displayed.



"**Open For Browsing**" invokes the MechWorks PDM user interface.

"**View Data**" displays the data record for the component that is currently in session.

"**Document Wizard**" filters by categories and it is through a graphical interface.

MechWorks PDM' "**Save As**" same as SolidWorks' "Save As" except it creates a record in the database.

MechWorks PDM' "**Save**" creates a database record in MechWorks PDM if the current file doesn't already have a record.

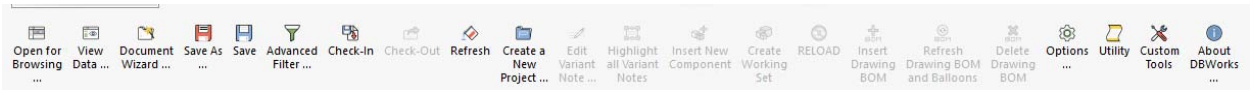
"**Advanced Filter**" allow users to filter the database to locate specific records and it is through a graphical interface.

"**Check-In**" allow users to "Check In" a component and "Save" it at the same time.

Checkout allow users to "Checkout" a component from within SolidWorks.

Refresh" is different than the SolidWorks refresh; allows users to refresh variant notes as well as pass along any changes that have been made to an open assembly. EXAMPLE: *"USER_A" has checked out a part and made changes to the part, while "USER_B" has an assembly that contains "USER_A's" part open. This Refresh icon can be used to notify "USER_B" that changes have been made to a part contained within their in-session assembly. "USER_B" can then decide to incorporate the updated part or to ignore the notification.*

"**Create A New Project**" allow users who have this privilege to create a new project.



”**Edit Variant Notes**” allow users to edit variant notes.

”**Highlight all Variant Notes**” allow users to highlight variant notes. A variant note is a note that is linked to the database and is automatically updated from the database.
EXAMPLE: *Part_number and/or Part_Name within the Title Block.*

”**Insert New Component**” allow users to insert a new component into an assembly using the ***Top Down*** method. Creating a new “Part” or an “Assembly” and enters its record into the database.

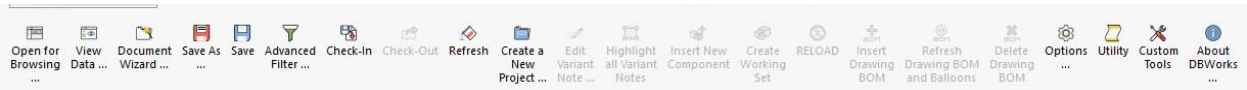
”**Create Working Set**” allow users to create a working set (group of documents) from SolidWorks.

“**Reload**” allow users to reload components that have been modified by other users (ignoring the “Refresh” command performed by the other users will activate this icon).

”**Insert Drawing BOM**” allows users to insert a MechWorks PDM BOM into the drawing.

”**Refresh Drawing BOM**” allow users refresh the MechWorks PDM BOM in the drawing.

”**Clear Drawing BOM**” allow users remove the MechWorks PDM BOM.



”**Options**” allow users to access the MechWorks PDM Options.

”**Utility**” allow users to access several utility commands such as the Activate Debug Logging.

”**Custom Tools**” allow administrators to install custom utilities and then users can run those custom utilities from the “**Custom Tools**” icon.

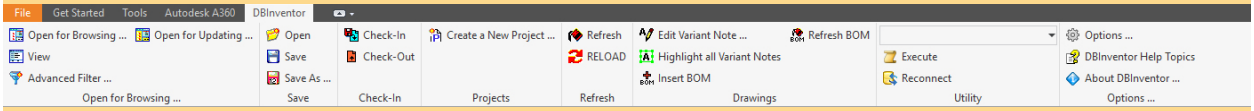
”**About DBWorks**” allow user to access the About DBWorks which displays information about the current DBWorks environment including the build of software that is installed.

With DBInventor activated, Inventor user's interface will display two areas of change:

- DBInventor "Ribbon Toolbar"
- DBInventor "Browser Bar"

Ribbon Toolbar

The Ribbon Toolbar is compliant to Inventor standards. As with other Inventor Ribbon Toolbars, the commands are grouped into different categories.




"**Open For Browsing**" invokes the DBInventor user interface.


"**Open For Updating**" invokes the DBInventor user interface in Update mode.


"**View Data**" displays the data record for the component that is currently in session.


"**Advanced Filter**" allows users to filter the database to locate specific records and it is through a graphical interface.


"**Open**" filters by categories and it is through a graphical interface.


DBInventor "**Save**" creates a database record in DBInventor if the current file doesn't already have a record.

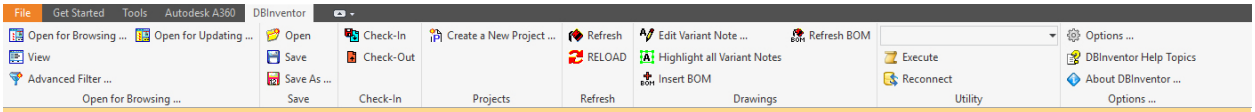

DBInventor "**Save As**" same as Inventor' "Save As" except it creates a record in the database.


"**Check-In**" allows users to "Check In" a component and "Save" it at the same time.


"**Checkout**" allows users to Check-out a component that is opened in Inventor.


"**Create A New Project**" allows users who have this privilege to create a new project.


"**Refresh**" is different than the Inventor refresh. Allows users to pass along any changes that have been made to an open assembly. EXAMPLE: *"USER_A" has checked out a part and made changes to the part, while "USER_B" has checked out an assembly that contains "USER_A's" part. This Refresh icon can be used to notify "USER_B" that changes have been made to a part contained within their in-session assembly. "USER_B" can then decide to incorporate the updated part or to ignore the notification.*



"**Reload**" allows users to reload components.

"**Edit Variant Notes**" allows users to edit variant notes.

"**Highlight all Variant Notes**" allows users to highlight variant notes. A variant note is a note that is linked to a database field and is automatically updated from the database. *EXAMPLE: Part_number and/or Part_Name within the Title Block.*

"**Insert BOM**" allows users to insert a MechWorks PDM BOM into an Inventor drawing.

"**Refresh BOM**" allows users to refresh the MechWorks PDM BOM that is in the Inventor drawing.

"Execute" allows users to execute any script from the drop down list above the icon.

"**Reconnect**" allows users that have lost their connection to the MechWorks PDM database to reconnect once the network connection has been reestablished.

"**Options"** allows users to access the DBInventor Options. Depending on the versions of DBInventor these may be Global Options or User Options.

"DBInventor Help Topics" allows users to access the DBInventor Help file.

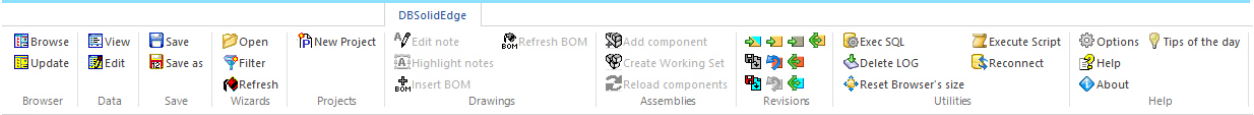
"About DBInventor" allows users to access the About DBInventor screen that displays helpful information about DBInventor including the build of software installed.

With DBSolidEdge activated, Solid Edge user's interface will display two areas of change:

- DBSolidEdge "Quick Access Toolbar"
- DBSolidEdge "Command Bar"

Quick Access Toolbar

The Quick Access Toolbar is compliant to Solid Edge standards. As with other Solid Edge Quick Access Toolbars, the commands are grouped into different categories.




"**Open For Browsing**" invokes the DBSolidEdge user interface.


"**Open For Updating**" invokes the DBSolidEdge user interface in Update mode.


"**View Data**" displays the data record for the component that is currently in session.


"Edit Data" displays the data record for the component that is currently in session in Edit Mode.


DBSolidEdge "**Save**" creates a database record in DBSolidEdge if the current file doesn't already have a record.

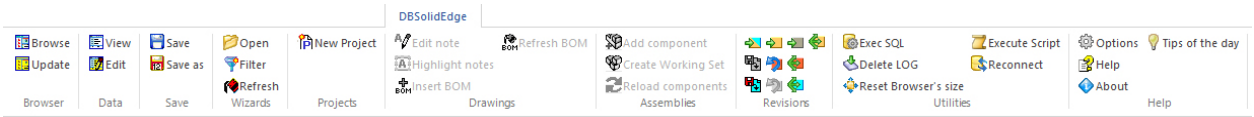

DBSolidEdge "**Save As**" same as Solid Edge' "Save As" except it creates a record in the database.


"**Open**" filters by categories and it is through a graphical interface.


"**Advanced Filter**" allows users to filter the database to locate specific records and it is through a graphical interface.


"**Refresh**" is different than the Solid Edge refresh. Allow users to refresh variant notes as well as pass along any changes that have been made to an open assembly. EXAMPLE: *"USER_A" has checked out a part and made changes to the part, while "USER_B" has checked out an assembly that contains "USER_A's" part. This Refresh icon can be used to notify "USER_B" that changes have been made to a part contained within their in-session assembly. "USER_B" can then decide to incorporate the updated part or to ignore the notification.*


"**New Project**" allows users who have this privilege to create a new project.



”**Edit Variant Notes**” allows users to edit variant notes.

”**Highlight all Variant Notes**” allows users to highlight variant notes. A variant note is a note that is linked to the database and is automatically updated from the database. EXAMPLE: *Part_number and/or Part_Name within the Title Block.*

“**Insert BOM**” allows users to insert a MechWorks PDM BOM into a Solid Edge drawing.

“**Refresh BOM**” Allows users to refresh a MechWorks PDM BOM that is in a Solid Edge drawing.

”**Add Component**” allows users to insert a new component into an assembly using the ***Top Down*** method. Creating a new “Part” or an “Assembly” and enters its record into the database.

”**Create Working Set**” allows users to create a working set (group of documents) from Solid Edge.

“**Reload**” allows users to reload components that have been modified by other users (ignoring the “Refresh” command performed by the other users will activate this icon).

“Increment Revision” allows users to perform an Increment Revision command against the in session document.

“Check Out” allows users to perform a Check Out command against the in session document.

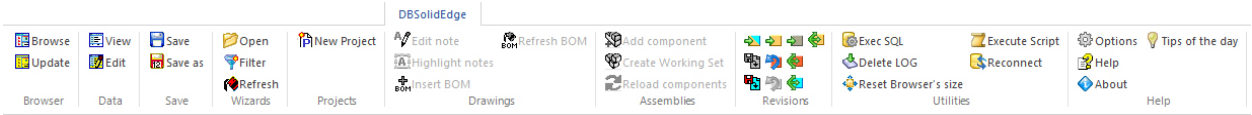
“Undo Check Out” allows users to perform an Undo Check Out command against the in session document.

“Check In” allows users to perform a Check In command against the in session document.

“Undo Check In” allows users to perform an Undo Check In command against the in session document.

“Approve” allows users to perform an Approve command against the in session document.

“Freeze” allows users to perform the Freeze command against the in session document.



"Recursive CheckOut" allows users to perform the Recursive CheckOut command against the in session document.

"Recursive CheckIn" allows users to perform the Recursive CheckIn command against the in session document.

"Recursive Approval" allows users to perform the Recursive Approval command against the in session document.

"**Execute SQL**" allows users to execute .SQL query files.

"**Delete LOG**" allows users to delete the Event Log.

"**Reset Browser's size**" allows users to reset the Browser back to the default layout.

"**Execute Script**" allows users to execute a .VBS script (such as Active Debug logging).

"**Reconnect**" allows users that have been disconnected from the MechWorks PDM to reconnect once the network connection has been reestablished.

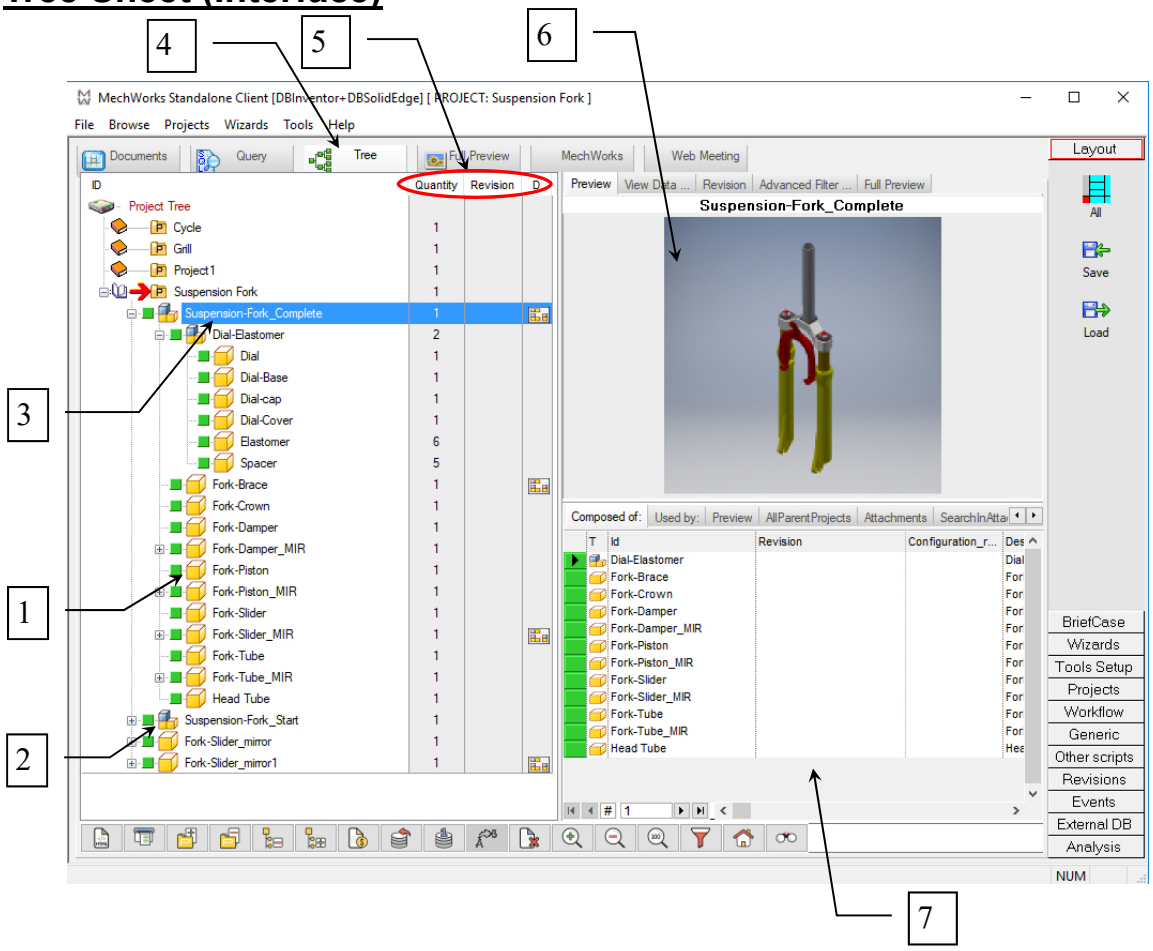
"**Options**" allows user to access the DBSolidEdge options.

"**Help**" allows users access to the Help file.

"**About**" allows users to access the **About DBSolidEdge**.

"**Tips of the day**" allows users access to the **Tips of the day** for DBSolidEdge.

Tree Sheet (Interface)



Once you select the "**Open for Browsing**" icon from the MechWorks PDM **Toolbar**, the MechWorks PDM User interface will be displayed and it will look similar to the above picture. The "**Tree**" interface is displayed and you will notice that the interface has logical grids (partitions). This figure represents the default layout but the user has complete control over the size and position of each of the grids.

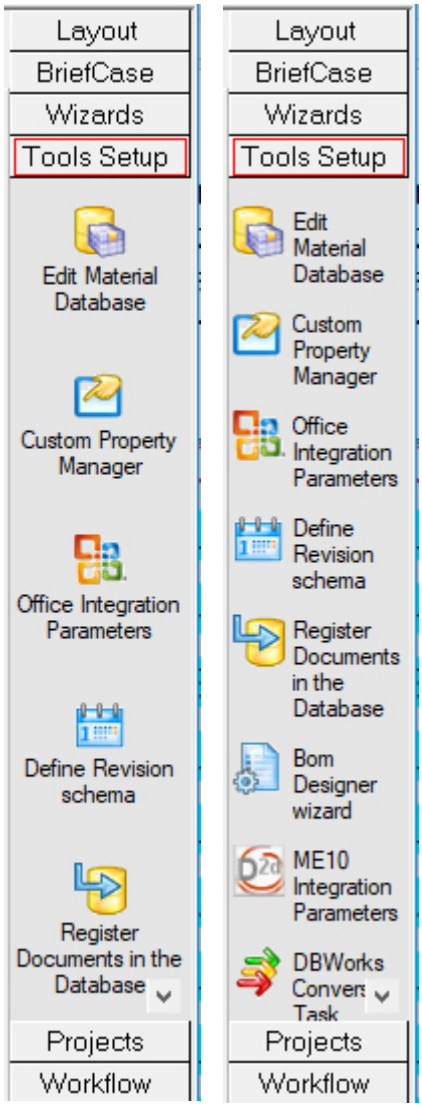
Tree Sheet User Interface

1. CAD **part** (icon) and "*STATE*" similar to the "**Project Tree Grid**".
2. CAD **assembly** (icon) and "*STATE*" (**hierarchical**) like the "**Project Tree Grid**".
3. Highlight current selection in the Tree Grid.
4. Tree Sheet (tab).
5. Customizable Columns
6. Selected Document's information (Preview, View Data, Revision, Advanced Filter & Full Preview).
7. Document Grids (Composed Of, Used By, Configurations, AllParentProjects, Attachments, Sheets, Search, Working Set, Part, Assembly, Drawing, Other Documents & Previous Revisions).

Shortcut Bar

The **shortcut** bar is fully customizable. It holds a collection of queries and scripts for easy access and logical groupings. Its look and feel is similar to Microsoft® Outlook's shortcut bar. The same **shortcut** bar is available as you navigate from the *Document sheet* to the *Query sheet* or to the *Tree sheet* for easy access to commonly used scripts. *NOTE*: The **shortcut** bar is an "*Option*" that may be turned on or off.

These two figures are just a few of the default *shortcut* bars available after you install MechWorks PDM.



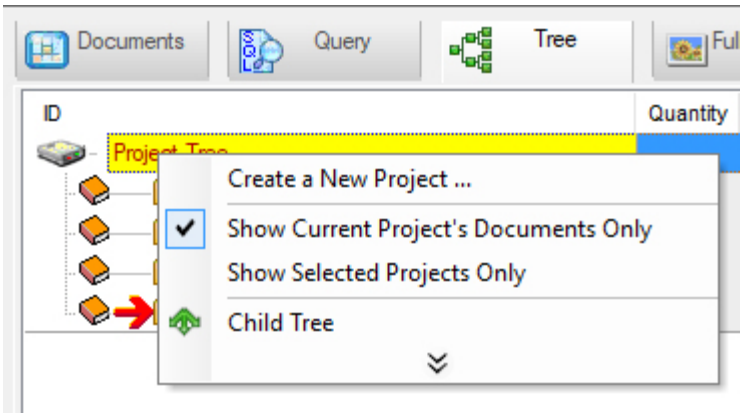
To see how these work, simply select some of the icons from the "**Layout**" *shortcut* bar. You will notice (while in the *Document Sheet*) that grid sizes and positions will change accordingly. Take a moment to review the specific icons found in the **Layout** *shortcut* bar. This will help you better predict the outcome.

The *shortcut* bar is often a tool that the system administrator will manage. The commands within each *shortcut* bar can be modified. You can also introduce new commands to a *shortcut* bar and remove existing commands from a *shortcut* bar.
It is also possible to add a new section or remove a current *section* from the shortcut bar.

TIP: If you **right** mouse click in any *shortcut* bar, a pop-up menu will appear allowing you two choices.

Context sensitive Pop-up Menu

MechWorks PDM has many *Pop-up* menus at its disposal when using your ***right*** mouse button. You should take the time to review the different menus from the different grids. Because these menus are context sensitive, commands may not be displayed depending on the type of document that you have selected (project/assembly/part/drawing, etc.). These *Pop-up* menus are also uniquely related to the grid and sheet that you are using.

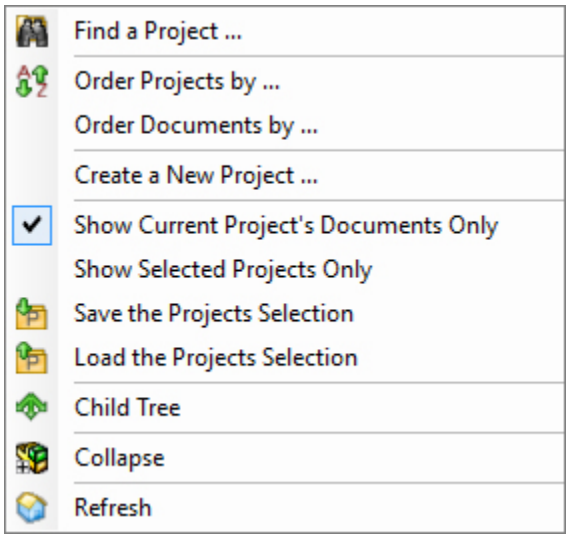


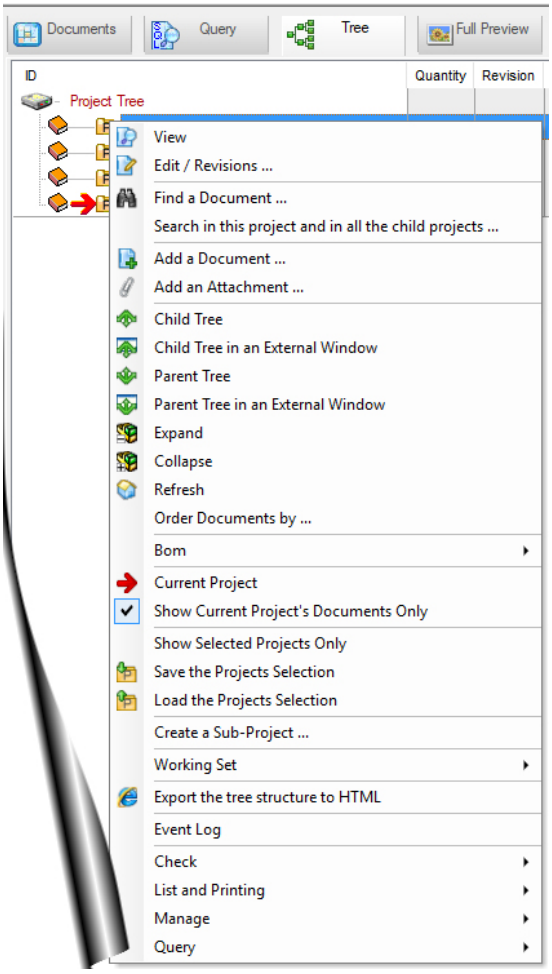
Project Root "*Pop-up*" menu.

The above menu is discussed in detail in the next lesson. For the purpose of viewing the different *Pop-up* menus, we do not currently need to know the commands within the menus. Notice the highlighted item just behind the *Pop-up* menu. From the above figure, you can identify that the image was taken from the Document sheet and the selection is the Project Root from the Project Tree Grid.

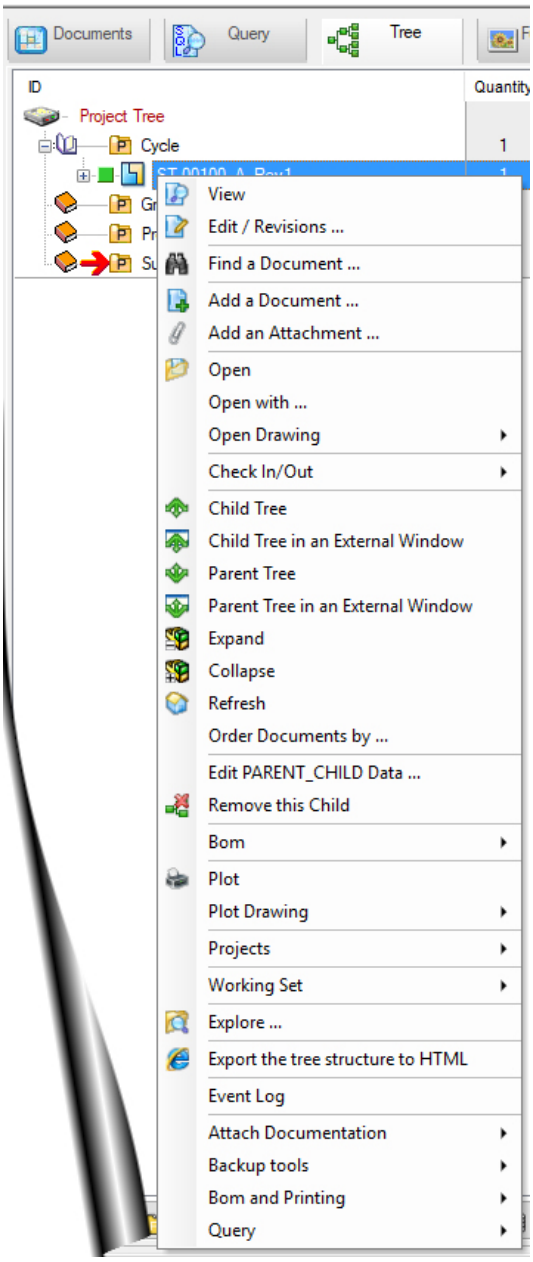
NOTE: Your **Option** settings can affect the display of the *Pop-up* menus. The above figure is using the following **Option**:

Whereas the figure on the right shows the same menu as above, when the same **Option** is deactivated:





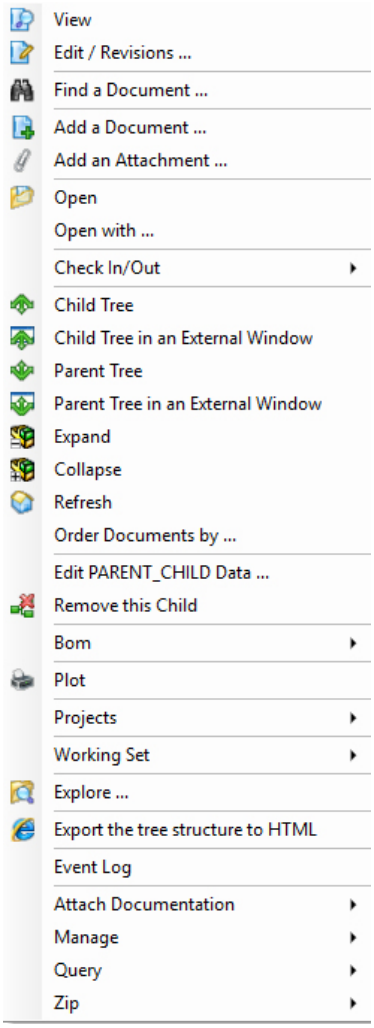
Project Pop-Up Menu



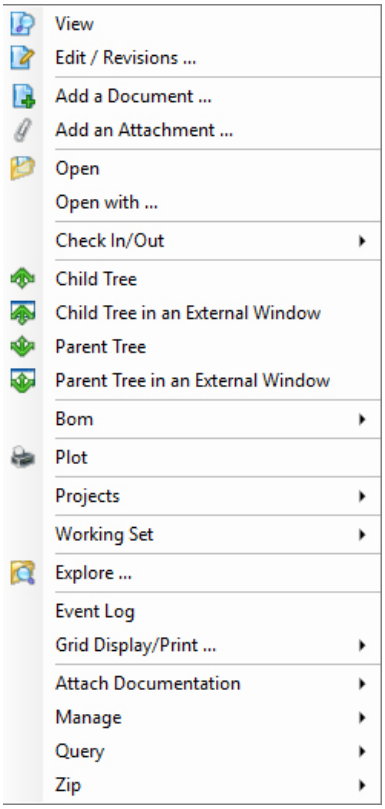
Assembly Pop-Up menu

By placing these two figures side by side, you get a good visual of the subtle differences that exist between *Pop-up* menus. This is because the *Pop-up* menus are context sensitive. Notice the highlighted item just behind the *Pop-up* menus. From the figure on the left, you can identify that the image was taken from the Document sheet and the selection is a Project from inside the Project Tree Grid. From the figure on the right, you can identify that the image was taken from the Document sheet and the selection is an Assembly from inside the Project Tree Grid.

As you can see, these two menus have some common commands available and many unique commands. As noted earlier, for the purpose of viewing the different *Pop-up* menus, we do not currently need to know the commands within the menus.



Part Pop-Up Menu (Project's Tree Grid)

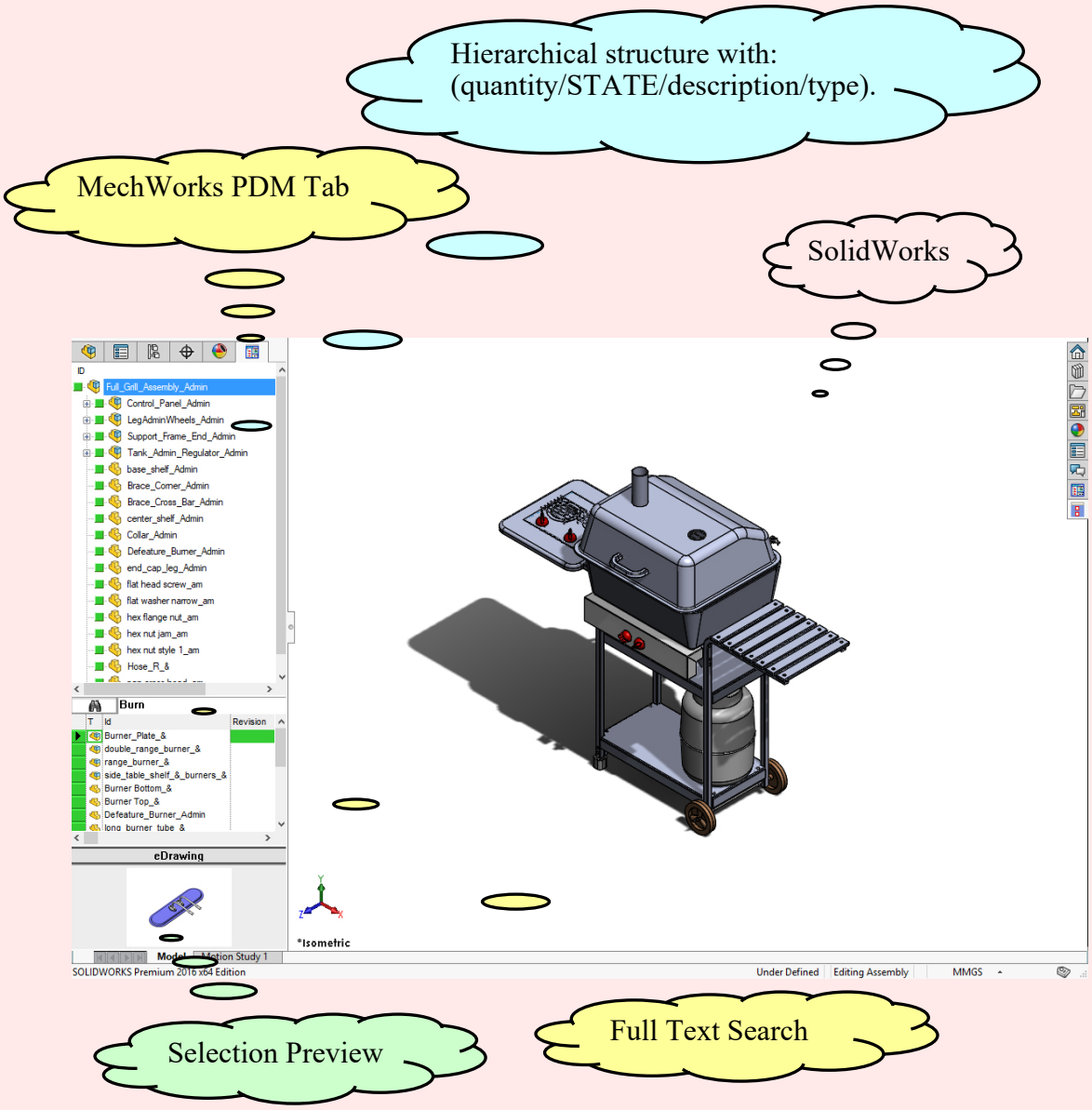


Part Pop-Up Menu (Part's Grid)

By reviewing the three figures on this page, you will better understand the differences in *Pop-up* menus. In both figures a "Part" was selected, yet the menus are different. Since all of these *Pop-up* menus represent a "Part", the differences for them have to be because of which grid they were generated from.

Feature Manager Window

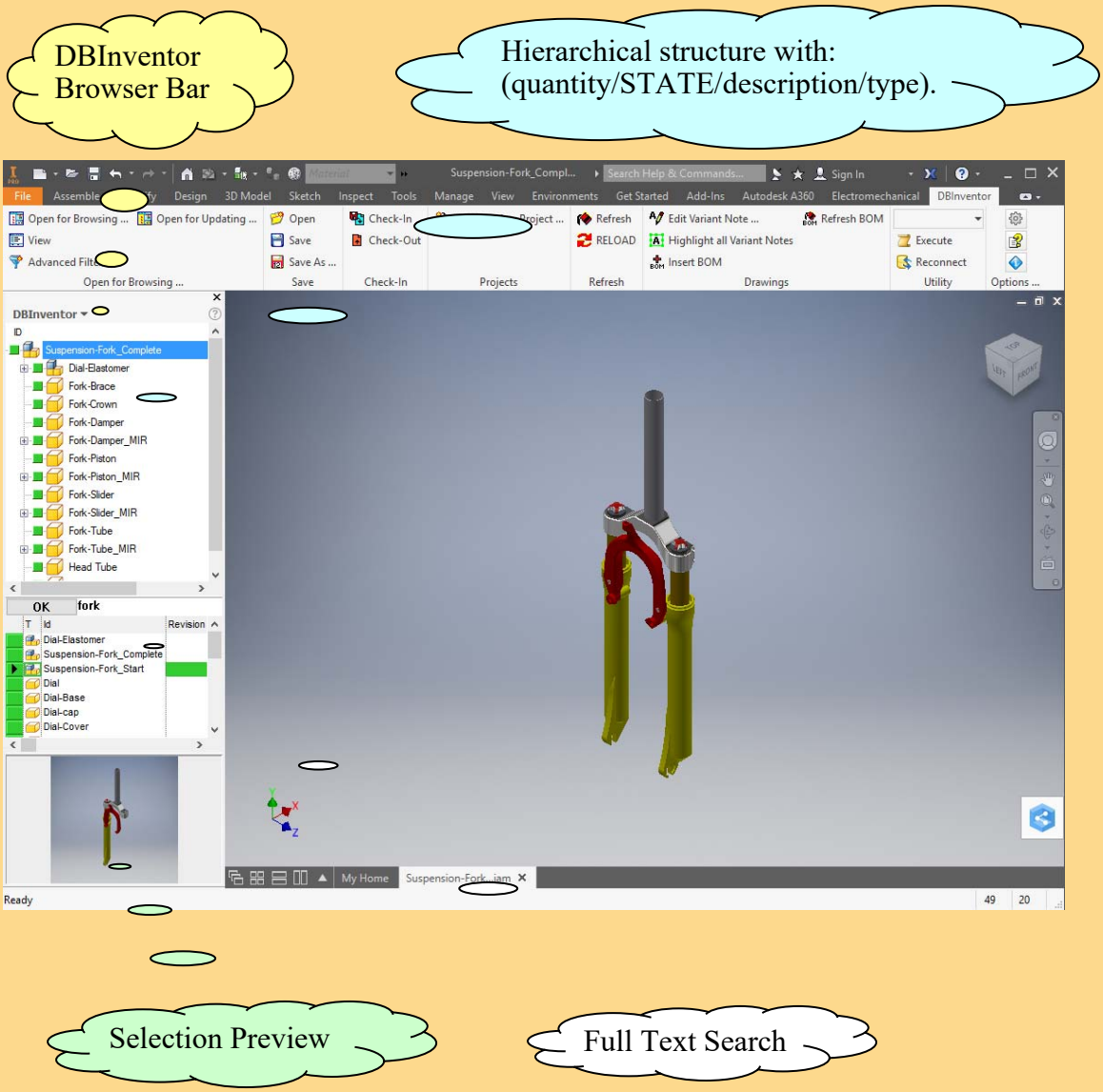
From SolidWorks, you have direct access to MechWorks PDM, through the MechWorks PDM tab that is added to SolidWorks' Feature manager. Because of this tight integration, users save time and effort carrying out standard MechWorks PDM tasks directly from SolidWorks.



The MechWorks PDM window contains the hierarchical structure. For assemblies and drawings, MechWorks PDM builds the child tree; for parts it builds the parent tree. The tree represents the component structure directly from the database; therefore, a "Save" or "Save As" is required to allow MechWorks PDM to update this windows hierarchy structure.

Inventor Browser

From Inventor, you have direct access to DBInventor, through the DBInventor Browser Bar that is added to Inventor Browser Bar. Because of this tight integration, users save time and effort carrying out standard DBInventor tasks directly from Inventor.



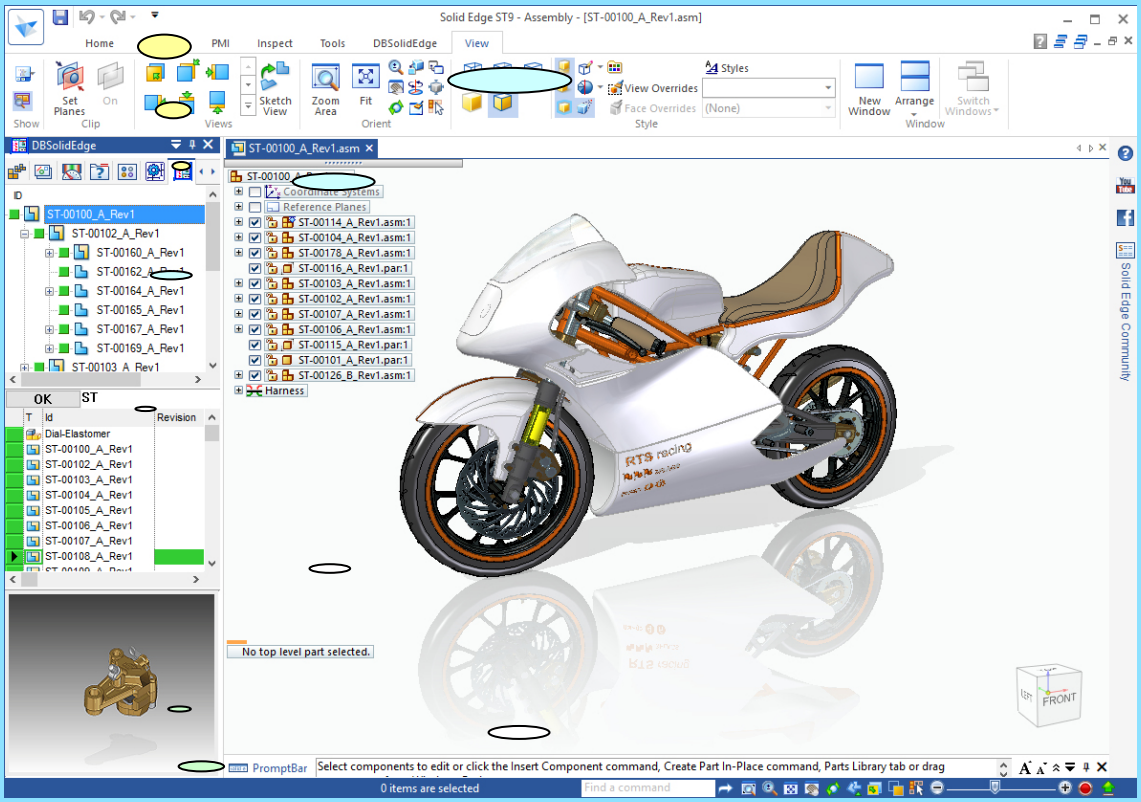
The DBInventor window contains the hierarchical structure. For assemblies and drawings, DBInventor builds the child tree; for parts it builds the parent tree. The tree represents the component structure directly from the database; therefore, a “Save” or “Save As” is required to allow DBInventor to update this windows hierarchy structure.

Solid Edge Browser

From Solid Edge, you have direct access to DBSolidEdge, through the DBSolidEdge Browser Bar that is added to Solid Edge Command Bar. Because of this tight integration, users save time and effort carrying out standard DBSolidEdge tasks directly from Solid Edge.

DBSolidEdge Command Bar

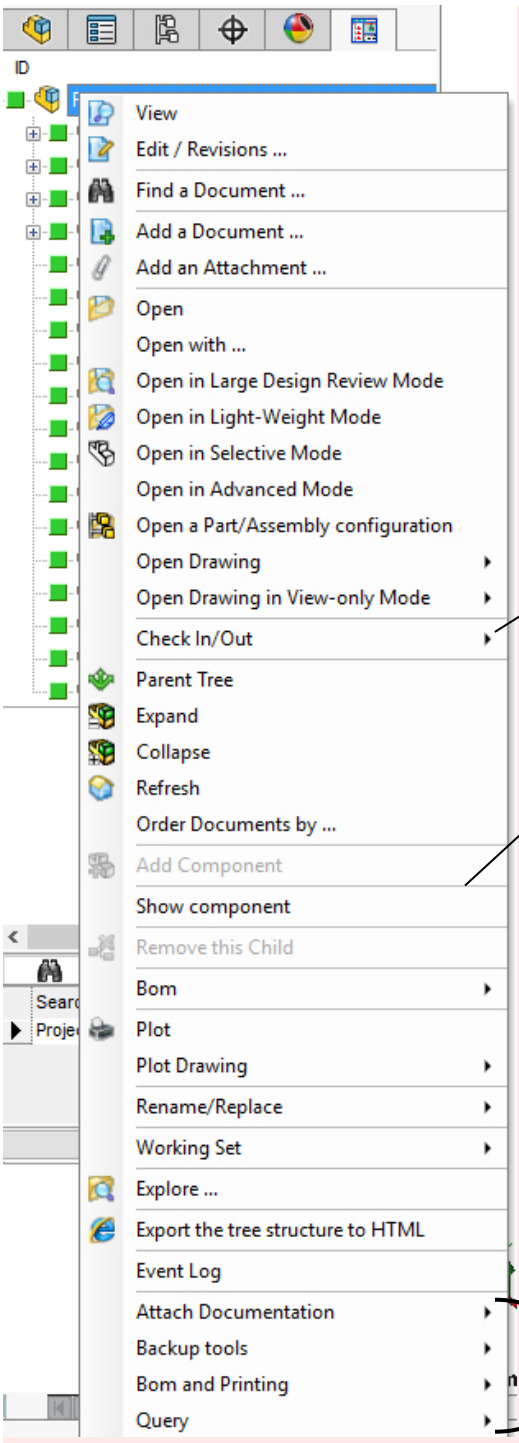
Hierarchical structure with: (quantity/STATE/description/type).



Selection Preview

Full Text Search

The DBSolidEdge window contains the hierarchical structure. For assemblies and drawings, DBSolidEdge builds the child tree; for parts it builds the parent tree. The tree represents the component structure directly from the database; therefore, a “Save” or “Save As” is required to allow DBSolidEdge to update this windows hierarchy structure.



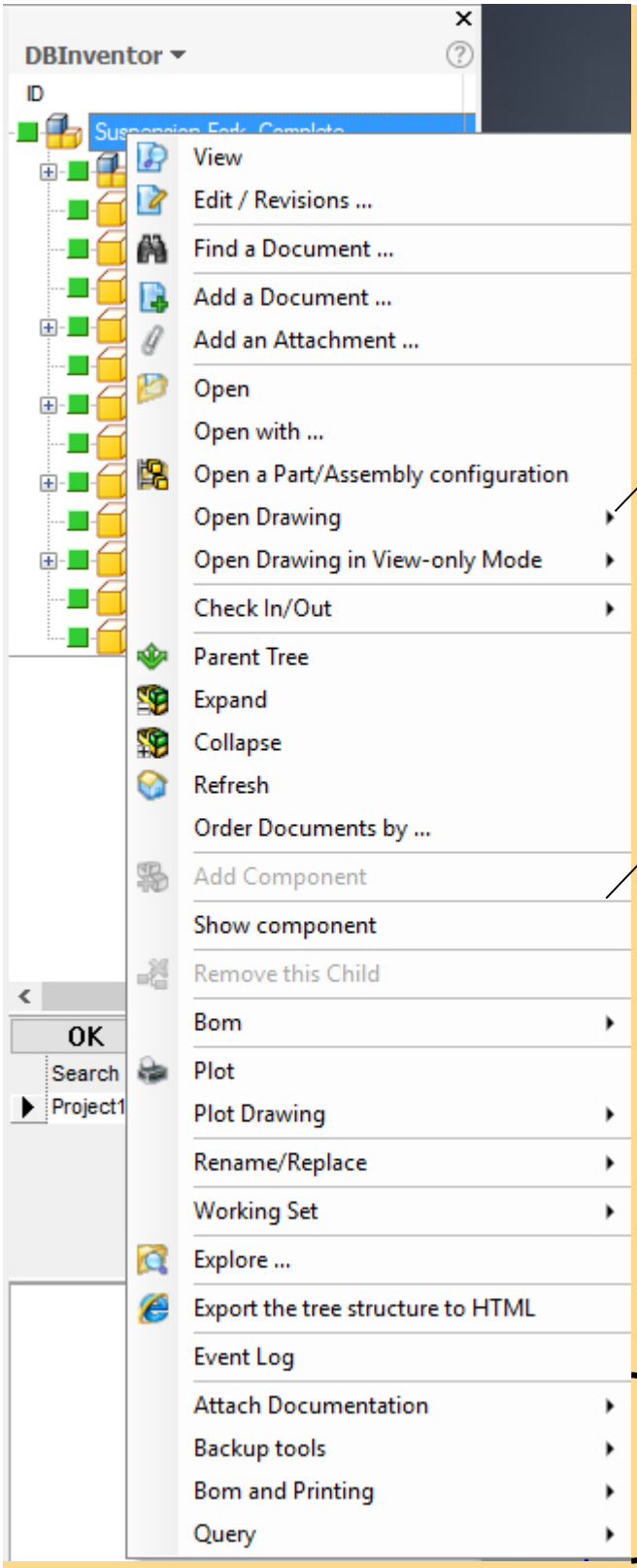
The *Pop-up* menu displayed in this figure is from the MechWorks PDM window in the SolidWorks' Feature manager. It is almost identical to the *Pop-up* menu that would be displayed from the **Tree** sheet.

This *Pop-up* menu contains most of the commonly required MechWorks PDM commands while working within a SolidWorks session..

(Arrow) represents sub-menu.

Section separation bar.

NOTE: The lowest section of a *Pop-up* menu can be easily customized by the administrator. Just like the menu itself, this section can also contain context sensitive commands.



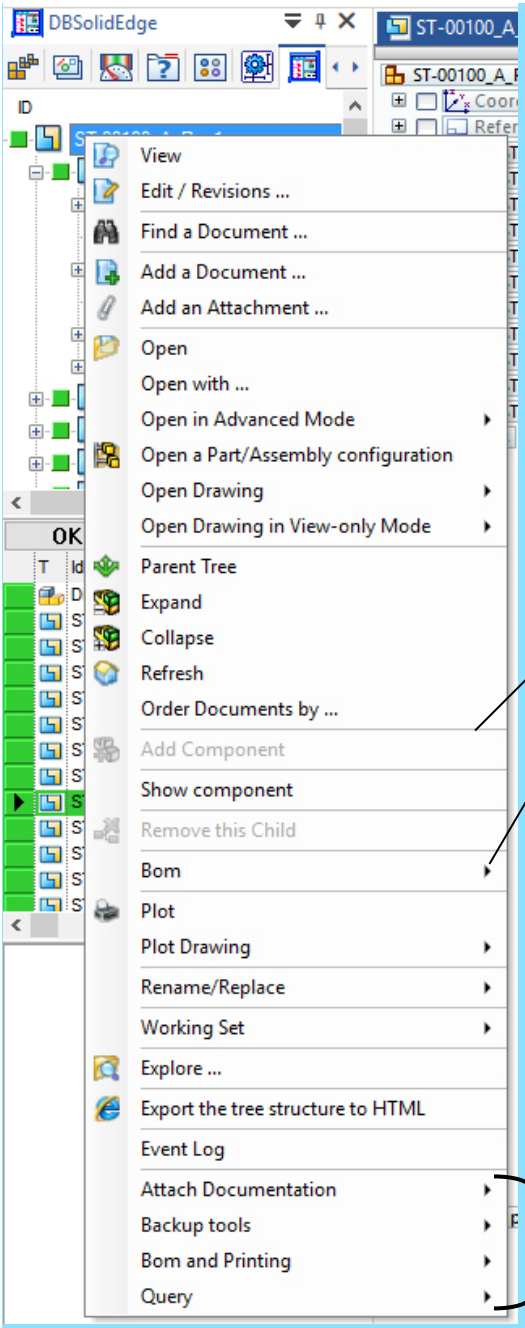
The *Pop-up* menu displayed in this figure is from the DBInventor Browser Bar in Inventor. It is almost identical to the *Pop-up* menu that would be displayed from the **Tree** sheet.

This *Pop-up* menu contains most of the commonly required DBInventor commands while working within an Inventor session.

(Arrow) represents a sub-menu.

Sectional separation bar.

NOTE: The lowest section of a *Pop-up* menu can be easily customized by the administrator. Just like the menu itself, this section can also contain context sensitive commands.



The *Pop-up* menu displayed in this figure is from the MechWorks PDM window in the Solid Edge Command bar. It is almost identical to the *Pop-up* menu that would be displayed from the **Tree** sheet.

This *Pop-up* menu contains most of the commonly required MechWorks PDM commands while working within a Solid Edge session..

Section separation bar.

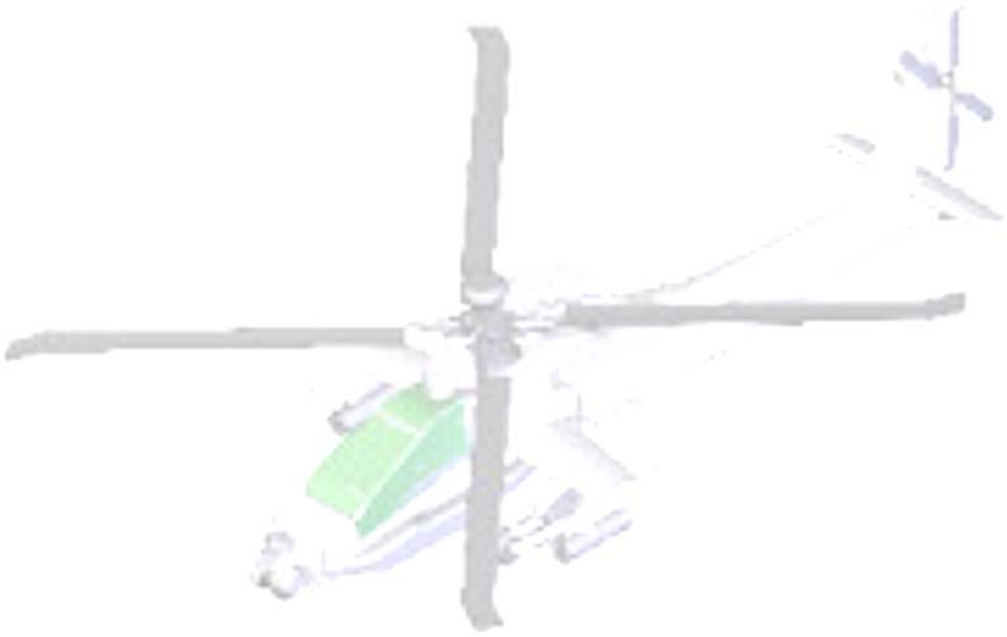
(Arrow) represents sub-menu.

NOTE: The lowest section of a *Pop-up* menu can be easily customized by the administrator. Just like the menu itself, this section can also contain context sensitive commands.


Intentionally Blank
as a separator

Chapter 5

Your first run with MechWorks PDM

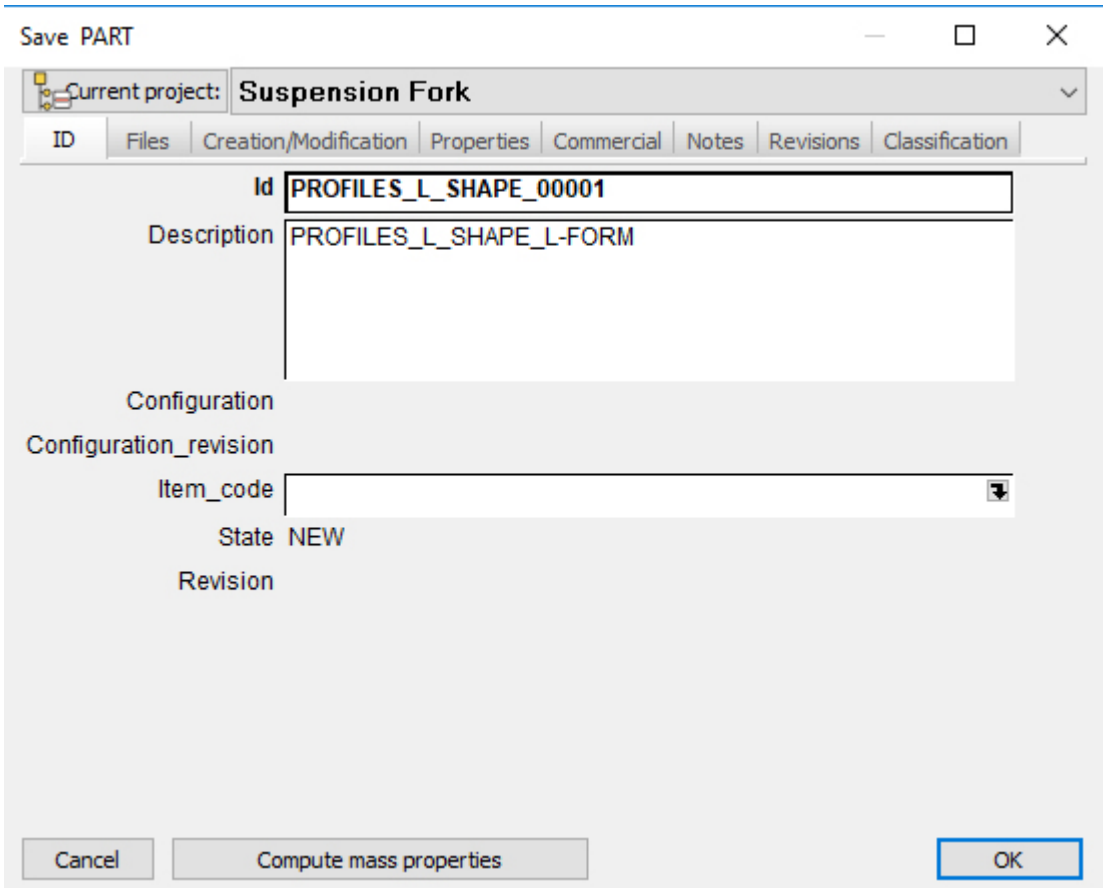


Your first run with MechWorks PDM

Now that you have installed MechWorks PDM and have had the opportunity to become a little more familiar with MechWorks PDM, it is time to begin working. In this chapter we will look at creating a new part file in SolidWorks and registering that part into MechWorks PDM. We will then look at taking an existing assembly (one of the demo assemblies included with MechWorks PDM) and registering the assembly and all the parts associated with this assembly into MechWorks PDM.

This example is done in SolidWorks but it could easily have been in Inventor or Solid Edge.

To begin, start a session of SolidWorks and create a simple part. With the part now completed, use the **File, Save** command. You will be asked categorize the new part (this is using a default *dataentr.lst* script). The MechWorks PDM data entry dialog will appear:

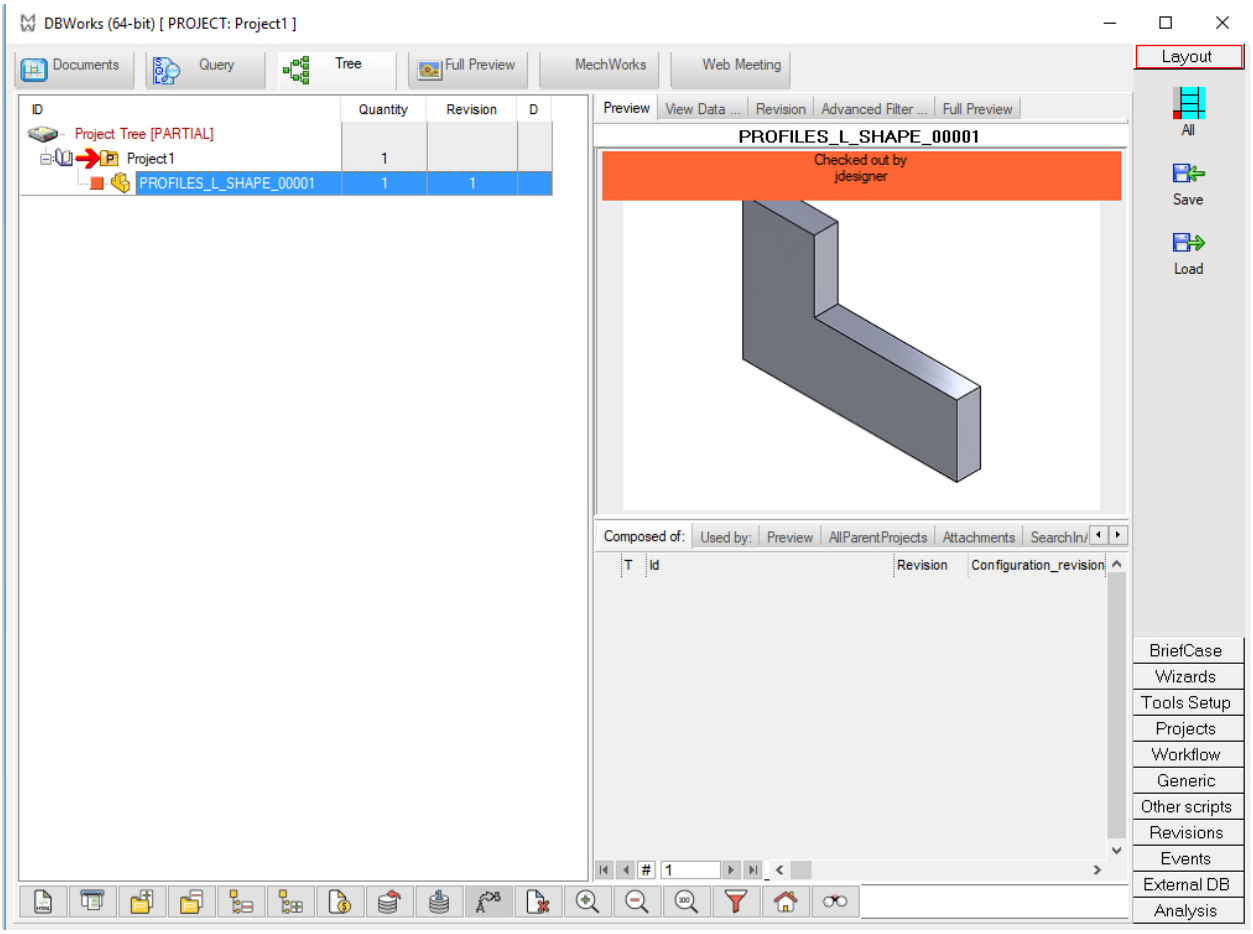


Click **OK**. Because, by default, the option to utilize the ***dataentr.lst*** script was used, the name and location of the file was automatically defined.

This will register this part in MechWorks PDM.

Now we will look at the MechWorks PDM Browser.

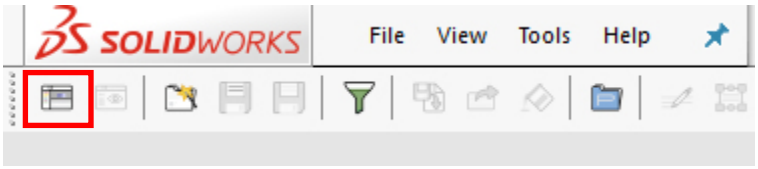
As you look at the MechWorks PDM Browser, you will notice the active *Project*, the record for the newly created part (***PROFILES_L_SHAPE_00001)*** and a preview of the newly created part.

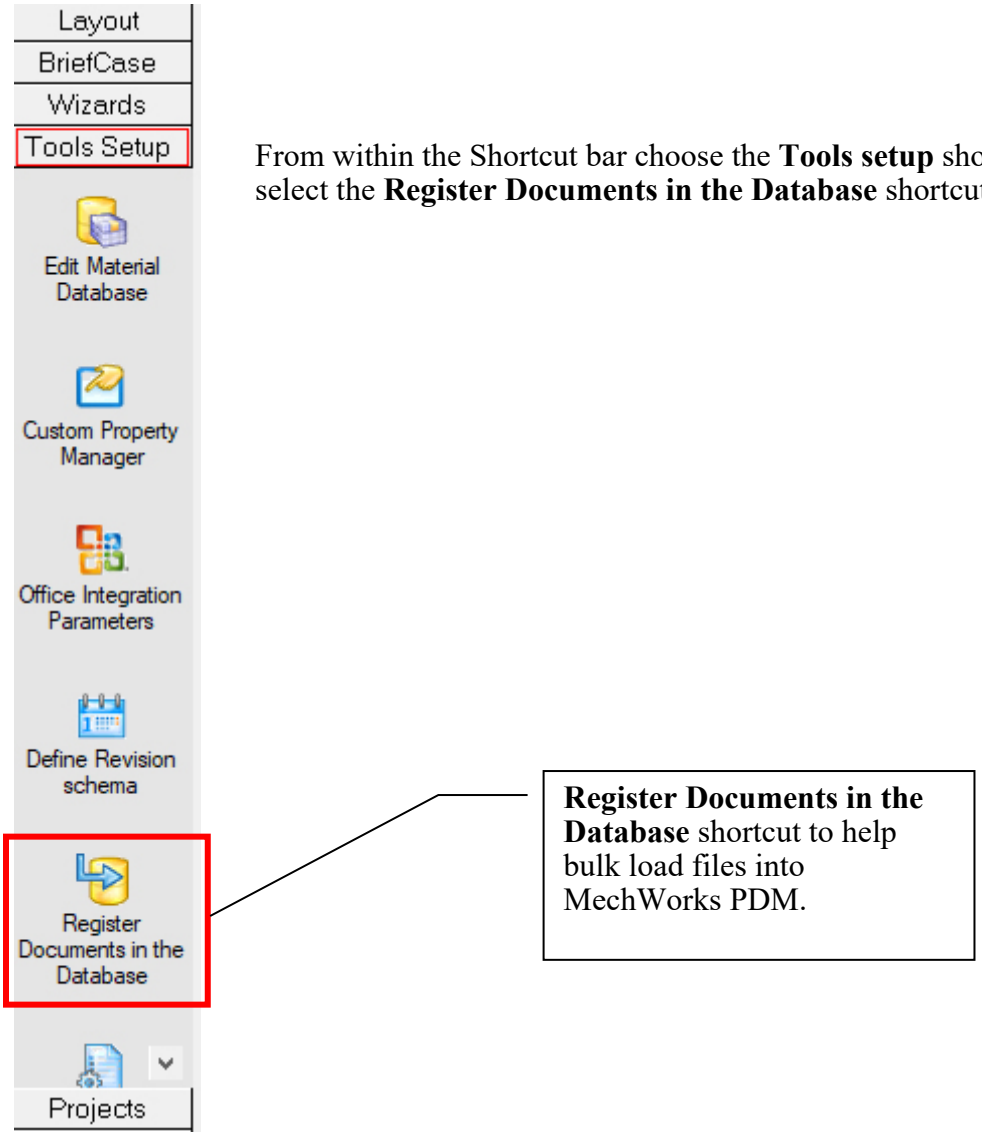


This is your first part registered in MechWorks PDM. Now we will look at registering an existing assembly.

Register existing documents into MechWorks PDM

From SolidWorks, choose the **Open for Browsing** command from the DBWorks toolbar.

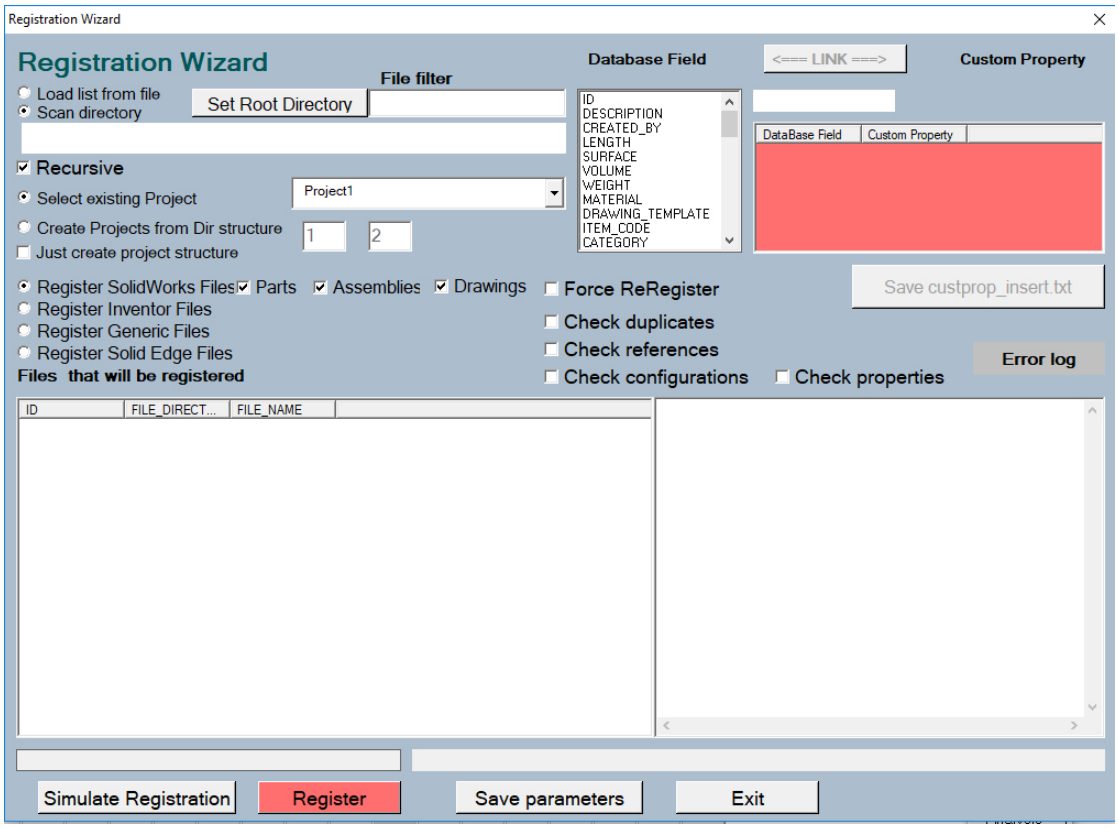




From within the Shortcut bar choose the **Tools setup** short cut group and select the **Register Documents in the Database** shortcut.

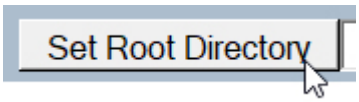
Register Documents in the Database shortcut to help bulk load files into MechWorks PDM.

Registration Wizard

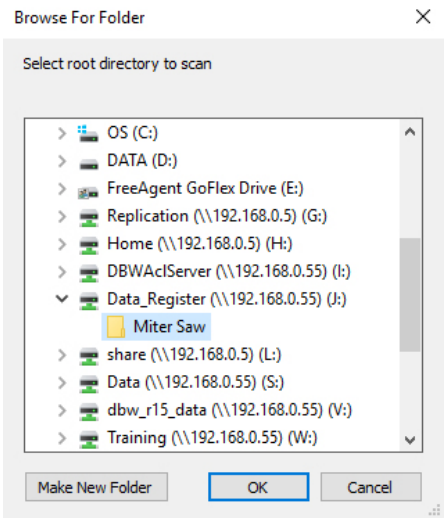


Choosing the **Register Documents in the Database** shortcut will display the **Registration Wizard** dialog. This wizard will help to bulk-load a folder of existing files into MechWorks PDM.

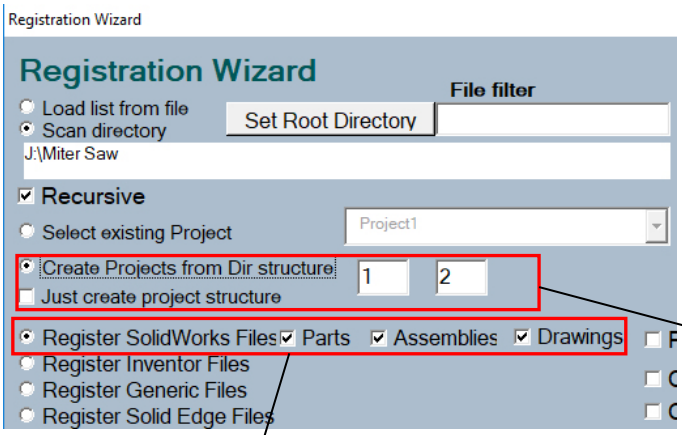
Left mouse select the **Set Root Directory** command.



Browse to the folder of files that you wish to register into MechWorks PDM.



After choosing the Root Directory, choose the option **Create Projects from Dir structure** and enter a project name for which these documents will be associated with. Also choose the **Register SolidWorks Files** option.

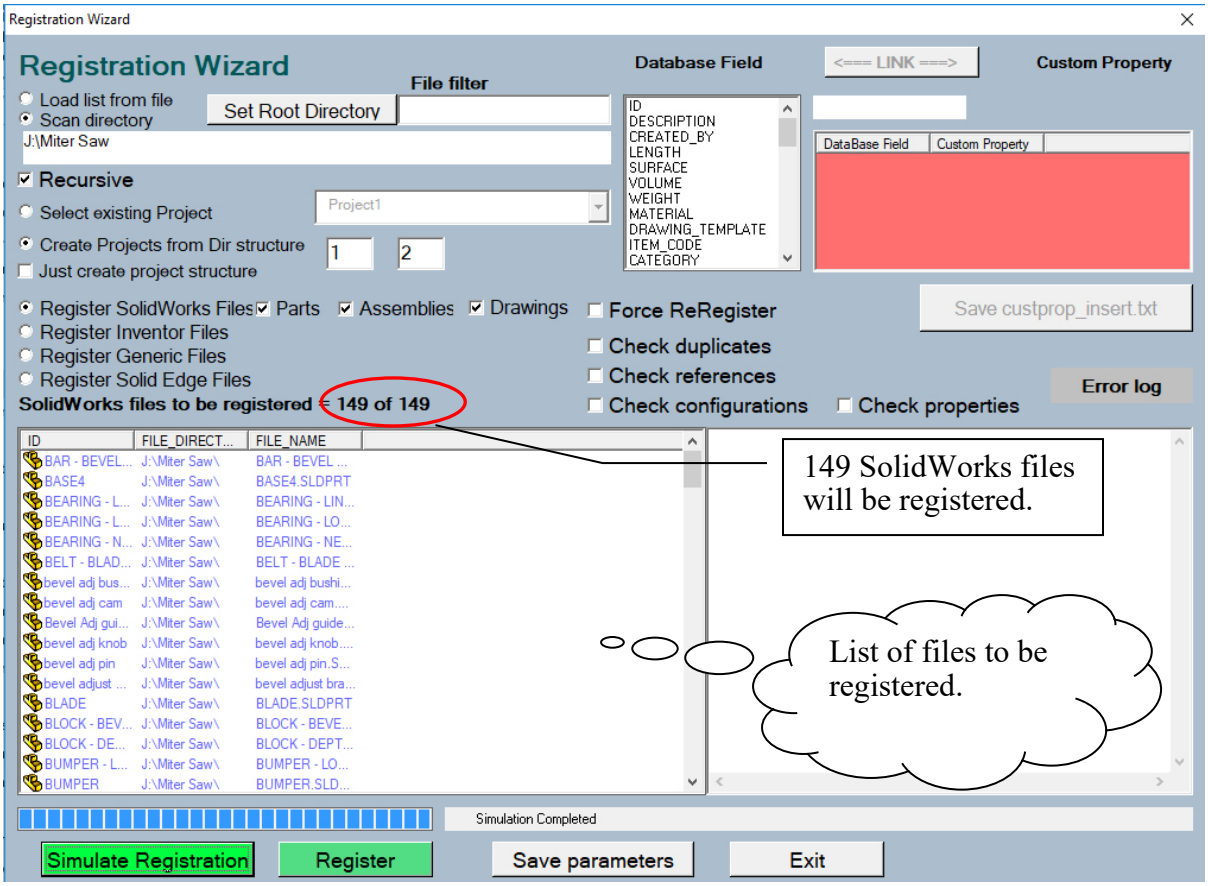


Select the **Register SolidWorks Files** option.

Select the **Create Projects from Dir Structure** option, assigning the starting and ending structure level.

With the root directory set, a new project name entered and the **Register SolidWorks Files** option set, left mouse select the **Simulate Registration** button.

Simulate Registration results



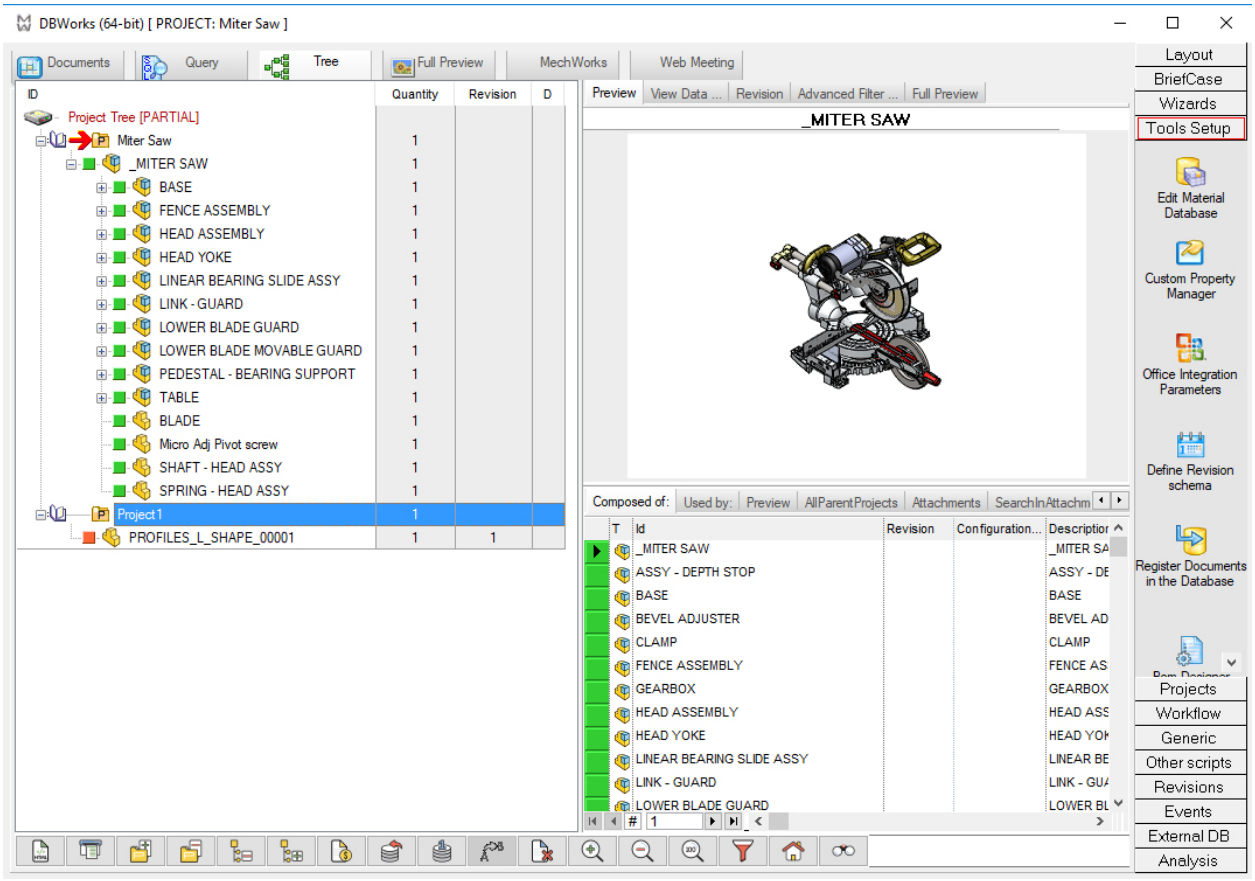
The results of the simulation are displayed within the wizard dialog. In this example there are 149 SolidWorks files that will be registered in MechWorks PDM.

To register the list of files, choose the **Register** button.

MechWorks PDM will now go through the process of registering each of the files within the list.

Note: This may take some time to finish. Each file is opened and saved within SolidWorks.

Once the **Registration Wizard** is complete, choose the **Exit** button. Open the MechWorks PDM Browser and notice that the new project is listed in the **Project Tree** and the files are displayed in their proper grids.



Registering some demonstration files allows users to become familiar with MechWorks PDM without having to work directly on production files.

MechWorks PDM offers several training courses, including Administrator and User training. For more information about MechWorks PDM, please contact your MechWorks PDM Reseller.


Intentionally Blank
as a separator