Intergraph Smart™ 3D

(Includes SmartPlant® 3D, SmartMarine® 3D, SmartPlant® 3D Materials Handling Edition)

Product: Smart™ 3D 2016 (Smart 3D)

Number: SEBY683AV-1100A (Purchase)

SEBY799AV-1100A (Lease)

Version: 2016 (11.00.84.0099)

To verify the Smart 3D version on Windows:

1. Click Control Panel > Programs and Features.
2. Enable the “Version” column on the window.
3. The Version column will display “11.00.84.0099” adjacent to Intergraph Smart 3D.

To check the component versions, run the VersionCheck.exe command from [*Smart_3D_Installation_Directory]\Core\Tools\Administrator\Bin*. Click the "Check All Task Versions" button. All the registered versions are displayed.

Date: November 18, 2016

Description: Smart™ 3D, the most advanced plant and offshore design software, is Intergraph® Process, Power & Marine’s data-centric, rule-driven solution for streamlining engineering design processes while preserving existing data and making it more usable/re-usable.

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System Requirements

Important: Beginning with Windows 10 and Oracle 12c Release 1 (12.1.0.2), Microsoft and Oracle will enforce the Internet Host Table Specification RFC 952 which mandates that component hostname labels can contain only alphanumeric characters. Hostnames using underscores ('_') are not allowed. Refer to Oracle Support Articles 1603775.1 and 1957895.1 and Microsoft KB 101785.

We highly recommend installing Smart 3D on a "clean" machine to avoid possible issues caused by incompatibility with other products. A "clean" machine refers to a fresh installation of Windows® 7, 8.1, or 10; or Windows® 2012, and the prerequisites as listed in the Installation Guide.

For more information regarding the hardware and software requirements for Smart 3D, please refer to the "Smart 3D Hardware and Software Recommendations" section of the Installation Guide.

Version Compatibility

For up-to-date information on the software compatibility of this product in a standalone or integrated environment, please refer to the Compatibility Matrix on the Intergraph PPM Support website at https://smartsupport.intergraph.com/.

Log on and perform the following steps:

1. Click the **View Downloads** tab.
2. Click the **Product Compatibility** link under **Useful Links** on the right side.
3. On the **PPM Compatibility Matrix - Product Report** page, from the Select Product list, select <Smart 3D>.
4. From the **Version** list, select the version of <Smart 3D>

Smart 3D 2016 was tested and is compatible with the following software:

Reference Data Server

- Microsoft Windows Standard/Datacenter Server 2012 R2 (64-bit)
- Microsoft SQL Server 2014 (64-bit)
- Oracle 12c Release 1 (12.1.0.2) (64-bit)
- Microsoft Internet Explorer 11
- Microsoft .NET Framework 4.5.2 or greater
- Adobe Acrobat Reader 11

NOTE: SQL databases in Smart 3D Global Workshare Configuration require the Enterprise Edition of SQL. Standard Edition is supported for stand-alone environments only.

NOTE: For Smart 3D Global Workshare Configuration with Oracle, Standard or Enterprise Edition of Oracle can be used.

Workstation

- Microsoft Windows 7, 8.1, or 10, Professional or Enterprise (64-bit) (see WINDOWS WORKSTATION NOTES below)
- Microsoft SQL Server 2014 Client Tools (64-bit)
- Oracle 12c Release 1 (12.1.0.2) Client Tools (32-bit)
- Oracle 12c Release 1 (12.1.0.2) Client Tools (64-bit) - only applicable with Smart 3D 64-bit services (IFC and Drawings Batch)

- SmartPlant License Manager 2012 (v12.0) or greater
- Microsoft Office 2013 (32-bit) (see MICROSOFT OFFICE NOTE below)
- Microsoft Internet Explorer 11
- Microsoft XML Core Services (MSXML) 6.0 SP1
- Microsoft .NET Framework 4.5.2 or greater
- A compatible PDF reader
- PDS 2011 SE (v12.0) and PDS 2011 R1 SE (v12.1)
- Solid Edge Versions ST3, ST4 and ST5
- MicroStation 8 (Note: This is only required if 3D DGN output will be created.)
- Tekla Structures V21 with Smart 3D Link
- Leica Cyclone 9.0
- CloudWorx for Smart 3D 11.0
- Intergraph Batch Services version 06.01.09.00 if you plan to use the client for remote batch processing
- Citrix XenApp 7.6 (Note: Citrix XenApp can only be installed on Windows 2012 Server R2 64-bit OS, and as such, Windows 2012 Server R2 64-bit OS will act as a client in this case. Windows 2012 Server R2 64-bit OS can also be used to install an ancillary server, such as IFC, Drawings and/or Intergraph Batch Services.)

SMARTPLANT® SCHEMA COMPONENT NOTE FOR Smart 3D 2016:

It is recommended to use SmartPlant Schema Component 2016 06.00.00.0025. This version contains a number of significant fixes to the SmartPlant Schema Editor which is installed with SmartPlant Schema Component. To install SmartPlant Schema Component 2016 06.00.00.0025, select "Additional Software" and then select the "SmartPlant Schema Component Installation" option on the Smart 3D Installation dialog. Different SmartPlant Schema Component versions should not be used in the same project. If the project has workstations using an older version of SmartPlant Schema Component, the machines should either be updated with SmartPlant Schema Component 2016 06.00.00.0025, or installed with the same version of SmartPlant Schema Component on newly installed workstations.

In the SmartPlant Enterprise environment, Smart 3D 2016 is compatible with SmartPlant Foundation Versions 2009 R3 (04.03.*), 2009 R4 (04.04.*), 2014 (05.*), and 2016.

Note: In a SmartPlant Foundation Integrated environment, use the SmartPlant Schema Component delivered with the SmartPlant Foundation product. In a standalone (non-integrated environment) SmartPlant Schema Component is used to generate SmartPlant Review output and can be installed from the Smart 3D DVD.

SMARTPLANT® LICENSE MANAGER NOTE FOR Smart 3D 2016:

Smart 3D 2016 requires SmartPlant License Manager 2012 or greater for license management. If you run into problems with licenses, check the Error Messages and Troubleshooting appendix of the SmartPlant License Manager Installation and User's Guide, delivered in the Help folder on the SmartPlant License Manager product CD.

SMART 3D 2016 INTERNATIONAL CONFIGURATION:

Smart 3D 2016 supports a "homogeneous" configuration, which is an environment that only uses elements from a single locale. For example, a German customer running on a German operating system, using only German characters and German cultural conventions, is a fully supported homogeneous configuration. It is highly recommended that you install the locale-specific language version of Microsoft Office. For example, on a Japanese Operating System, you would install Japanese Microsoft Office products. Internationalization testing was not done on Citrix.

There are internationalization considerations concerning the SQL Server collation settings you specify that relate to reference data, using SmartPlant Enterprise integrated systems, Global Workshare Configuration, multiple locales, backup/restore, reports, and upgrading to future releases. Please contact Intergraph Customer Support for specific configuration questions.

SMART 3D 2016 DISPLAY NOTES:

There have been many enhancements to the graphic engine of Smart 3D 2016. Most issues are resolved by loading the most current display driver or rolling your current display driver back one version. If the current driver does not fix your particular card's problem, then please copy the "CoreDisplaySettings.ini" file
from <*Smart_3D_Installation_Directory*>\Core\Tools\Administrator\Bin to
<*Smart_3D_Installation_Directory*>\Core\Shared\Bin and contact your Smart 3D support representative.

Smart 3D recommends client display settings set to 100%. Use of Display Settings larger than 100% may result in distorted menus and text.

Intergraph recommends Workstation class graphics cards (designed for use with CAD applications) with as much RAM as is affordable. The minimum RAM is 256MB, but more RAM is much better. Since the world of graphics cards changes so quickly, we can only document the specifications required by Smart 3D.

Please note that if display issues are observed on a client referencing Point Cloud data, Smart 3D hardware should be disabled with the "CoreDisplaySettings.ini" file.

As a supplement to this information, a spreadsheet is provided on Smart Support that records anecdotal evidence obtained from users. Please logon to Smart Support at http://smartsupport.intergraph.com and select View Downloads -> Smart 3D -> Technical Notes and White papers -> Display -> Graphics cards used with Smart 3D.

Please note that the above list is not comprehensive. In some cases, the Smart 3D middle tier error log (with the error logging level set to at least "2-Detailed") will provide more information. Please see the examples of messages returned in the error log listed below:

- The graphics card or graphics card driver does not support Textures/Shaders. An updated graphics driver could improve performance.
    - This indicates that the graphics adapter, driver or display settings do not support the "Shaded with Hardware Enhanced Edges" Render Mode on the Format View dialog box. In addition, the graphics adapter, driver or display settings do not support textures.
- The graphics card or graphics card driver does not support Shaders. An updated graphics driver could improve performance.
    - This indicates that the graphics adapter, driver or display settings do not support the "Shaded with Hardware Enhanced Edges" Render Mode on the Format View dialog box. However, textures are supported.
- The graphics card or graphics card driver does not support Textures. An updated graphics driver could improve performance.
    - This indicates that the graphics adapter, driver or display settings do not support textures. However, the "Shaded with Hardware Enhanced Edges" Render Mode on the Format View dialog box is supported. Please note that the performance may be slow on an AMD card.
- Hardware Acceleration is Turned off VBO Texture and Hardware edges will not be supported.
    - The following message indicates that Hardware Acceleration in Display Settings is disabled or the client is being accessed via Remote Desktop or Citrix Client. This also applies to systems that use Intel chipset graphics acceleration, where hardware acceleration is disabled due to driver stability. If Hardware Acceleration in Display Settings is disabled, it should be enabled by selecting the "Advanced" button on the Settings tab of the Display Properties dialog. Then, the Hardware Acceleration can be modified from the Troubleshoot tab.

IMPORTANT: Due to the 32-bit precision limitations of graphic cards, you may need to create several design coordinate systems so that the objects being modeled are within 10,000 meters (6.2 miles) of the global coordinate system. Objects modeled outside this limit will not display correctly. If your model coordinate values are larger than this limit (for example, E = 20,000, N = 30,000), to get the coordinate readout that you want, you should define a coordinate system at correspondingly large negative values (example, E = -20,000, N = -30,000). Then, use the coordinate system that you created as your active coordinate system for modeling and output. Do not bring this new coordinate system into your workspace, or you will have the display problems that you are trying to avoid. In addition to the 10,000-meter global coordinate system limit described above, the marine mode is limited to a 5,000-meter cube.

WINDOWS WORKSTATION NOTES:

- Smart 3D 2016 was tested and is compatible with the Microsoft Windows 7, 8.1, and 10 Professional and Enterprise Client 64-bit operating systems. Please refer to the *InstallationGuide.pdf* delivered with Smart 3D 2016 for additional Windows workstation OS related information and specific installation instructions.
- The recommended User Account Control settings for workstations with Windows is "ON". For Windows workstations, the level 3 UAC setting (Default - Notify me only when programs try to make changes to my computer) should be selected. Smart 3D has been tested and certified with these UAC settings.

- In some cases, when Smart 3D is installed on a Windows workstation with UAC turned OFF and then UAC is turned ON after installing or when UAC is turned ON and installed without installing as “Run as Administrator”, users may receive errors when creating a Site database with the Database Wizard or when opening Project Management or a Smart 3D session. In this situation, the user can reinstall Smart 3D using the right click menu “Run as Administrator” option on the Setup.exe or the user can run the Database Wizard, Project Management, or Smart 3D session by opening with the right click menu “Run as Administrator” option.
- Users should install and configure SmartPlant License Manager (SPLM) before installing Smart 3D on a Windows operating system. This will ensure that the user does not have to "Run as Administrator" when connecting to a site or when opening Project Management or Smart 3D. Please note that the user should "Run as Administrator" when configuring SPLM on the client.

MICROSOFT OFFICE NOTES:

- Smart 3D is compatible with Microsoft Office 2013 32-bit only. Macros are not working when MS Office 64-bit is installed on Windows 7, 8.1, or 10 64-bit OS machines. Smart 3D has been tested and certified with Office 2013 (x86).
- Smart 3D will only function with Excel XLS and XLSX formats.

ORACLE CONFIGURATION NOTES:

- Smart 3D is a 32-bit application that uses the 32-bit Oracle Client even when loaded on a 64-bit client operating system.
- If your operating system is 64-bit (Windows 7, 8.1, or 10 x64; or Windows 2012 x64), then the possibility exists that Smart 3D is installed to the 'default' program file location. In a 64-bit operating system from Microsoft, the default location is "Program Files (x86)". Because of this, the Oracle client must be 12.1.0.2 in order to interact with applications that are installed on the system in a location that contains a parenthesis in the path.
- Do not use the Oracle "light client" as it does not include some assemblies required by Smart 3D.
- A 64-bit Name Generator service is available that can be installed on Oracle 64-bit database servers with 64-bit Oracle client installed. The option to select the 32-bit or 64-bit Name Generator is available only on 64-bit workstations, but the 32-bit Name Generator can be installed on both 32-bit and 64-bit workstations with 32-bit Oracle client installed. However, it is not recommended to install the 32-bit Name Generator on an Oracle 64-bit database server. This configuration would also require Oracle 32-bit Client Tools to be installed on the server, and this causes performance degradation.

VIRUS SCANNERS AND PERFORMANCE:

- Other programs running on the computer, such as virus scanners, media players, and mail programs, might affect the performance of Smart 3D. Virus scanners affect performance by continuously checking the data files and log files. Exclude the database data folder on the database server from the virus scanner. For maximum performance, reduce the number of processes running while using Smart 3D.

FIREWALL NOTES:

- Port 135 should be opened and an exception should be created for RPC Dynamic Ports. See the Smart 3D Installation Guide for more details.
- If you have a firewall or anti-virus program setup on your client machine, you need to add the SmartPlant License Manager executable, pdlice.exe, in the exception list. If you do not add it to the exception list, when you try to reserve a seat that was taken by a process which terminates abnormally, you may notice:
    - That the seat is not released by the reserving process, and
    - The reserving log file contains the error "Connection Time Out".
- For the Windows Firewall, pdlice.exe needs to be added to the exception list.
    - Perform the following steps for Windows operating systems:
        - Click **Control Panel > Windows Firewall**.
        - Click **Allow a program or feature through Windows Firewall**.
        - Click **Change settings**.
        - Select **Allow another program**.
        - In the **Browse** dialog box, find the location of the ***pdlice.exe*** program (the default location is: c:\win32app\ingr\pdlice\bin or c:\win32app\ingr\splm\bin) in the **File name** box.
        - Click **Open**.
        - Click **Add** on the **Add a Program** dialog box.
        - Verify that ***SmartPlant License Manager*** has been added to the Allowed programs and features list.

Installation

Please note that the removal of any previous version of Smart 3D, SmartPlant 3D®, SmartMarine 3D® or SmartPlant 3D® Materials Handling Edition is **required** before installing Smart 3D 2016.

Refer to the Smart 3D Installation Guide for instructions on how to install and configure the software. This guide can be found in the Help folder on the installation DVD. It is also available post-installation at <*Smart_3D_Installation_Directory*>\Documentation\Help.

Intergraph Smart 3D 2016 has one product DVD which includes the following Products & Features: Smart 3D Installation, Reference Data Installation, Name Generator Installation(s), Smart 3D PDS Model & Data Translators Installation, Smart 3D Tribon Interface Installation, and optional Programming Resource installation software. The product DVD also includes the optional PDS Export installation software, SmartPlant Foundation Schema Component, and additional Smart 3D documentation.

Note: PDS Data Access tool is no longer a separate install. It has been combined with the PDS Export Tools Installation.

To install **Smart 3D**, use the ***Smart 3D*** serial number delivered with the Smart 3D DVD.

To install **Smart 3D PDS Model & Project Translators** and **Smart 3D Tribon Interface,** use the ***Smart 3D Translators*** serial number delivered with the Smart 3D DVD.

To install **PDS Export Tools**, use the ***PDS*** serial number delivered with the PDS CD.

To install **Intergraph Batch Services**, run the Setup.exe file in the Batch Services folder located on the Batch Service media. Intergraph Batch Services is delivered on its own CD that comes with your Smart 3D product DVD. For other installation details, see *Installing Intergraph Batch Services* in the Smart 3D Installation Guide.

Smart 3D 2016 requires the SmartPlant License Manager 2012 or greater software for concurrent licensing for Smart 3D. This licensing software is delivered on its own CD that comes with your Smart 3D product DVD. For more information about using and configuring concurrent licensing, see the SmartPlant License Manager Installation and User's Guide.

It is recommended to use SmartPlant Schema Component 2016 06.00.00.0025. This version contains a number of significant fixes to the SmartPlant Schema Editor which is installed with SmartPlant Schema Component. To install SmartPlant Schema Component 2016 06.00.00.0025, select "Additional Software" and then select the "SmartPlant Schema Component Installation" option on the Smart 3D Installation dialog. Different SmartPlant Schema Component versions should not be used in the same project. If the project has workstations using an older version of SmartPlant Schema Component, the machines should either be updated with SmartPlant Schema Component 2016 06.00.00.0025, or installed with the same version of SmartPlant Schema Component on newly installed workstations.

Upgrading to 2016

There are several steps involved in upgrading your configuration to Smart 3D 2016. Smart 3D 2016 supports upgrade from the following Smart 3D Versions:

- Version 2014
- Version 2014 R1

For detailed steps and explanations, please review the 2016 Upgrade Guide located in the Help folder on the Smart 3D 2016 product DVD.

For upgrading from version 2014 or 2014 R1 to version 2016, databases in a Global Workshare Configuration must be consolidated before upgrading.

Documentation

General

Use the Help menu to access the Help files and Printable Guides for this product. To suggest a change or to ask a question about the documentation, send an e-mail message to PPMdoc@intergraph.com

Printed documentation is not available for separate purchase.

The most recent guides and online Help are in the Help directory on the DVD. Copy the updated files from the Help directory on the DVD to the <*Smart 3D_Installation_Directory*>*Documentation**Help* folder on the client machines.

PDF Reader

The documentation is provided as PDF files. To view the PDF files, you must install a PDF reader.

Note: In some operating systems, Internet Explorer blocks the PDF files as active content. You must click the Information Bar, click "Allow Blocked Content", and then click "Yes" when asked if you are sure you want the file to run as active content. You must do this for each PDF file that you open. You have two options to prevent Internet Explorer from blocking PDF files.

- Option 1: Configure the PDF software to display PDF files in the native application, not in an Internet Explorer window, thus bypassing the need for an active content check. To do this, disable the option to Display PDF in Browser Option in the application options.
- Option 2: Change your Internet Options to allow active content (the PDF files) located on your computer to open. To do this:
    - In Internet Explorer, click Tools > Internet Options.
    - Select the Advanced tab.
    - Scroll to the Security section of the settings.
    - Select the Allow active content to run in files on My Computer option.

Training

To register for training on Intergraph Process, Power & Marine products, call Training Registration at (800) 766-7701 in the U.S. Outside the U.S., call (256) 730-5400 or contact your local Intergraph office.

For current information on training, use a World Wide Web browser to connect to http://www.intergraph.com/ppm/training/.

Customer Support

For the latest Support Services information for this product, including solutions to known software issues, use an Internet browser to connect to http://support.intergraph.com/http://support.intergraph.com/.

To open service requests outside the U.S., please contact your local Intergraph office.

Helpful Hints

Setup

- All major versions (**00**.00.00.00) and point release versions (00.**00**.00.00) of Smart 3D require the removal of all previous versions. For example, you cannot install Smart 3D 2016 over version 2014 R1 or version 2014. These versions, as well as any other service packs or hotfixes, must be removed before installing Smart 3D 2016.

- Follow the procedures in the *Smart 3D Installation Guide* when configuring the machines. Installing other software and/or deviating from the loading order may cause unexpected results. Do not skip any recommended reboots.

- If you have not already installed and configured SmartPlant License Manager on the workstation, we recommend doing so before installing Smart 3D.

- Silent Install/Removal scripts **must** be "Run as Administrator" on Windows 7, Windows 8.1, and Windows 10 clients. Please also note that Silent Install/Removal features that are input on the command-line are case-sensitive.

- To access an instance of Microsoft SQL Server 2014 through the Windows Server 2012 firewall, you must configure the firewall to allow access by performing the following steps:
    - Open the Control Panel and select Security > Windows Firewall. Select Advanced Settings.
    - Under "Windows Firewall with Advanced Security on the Local Computer", right click on Inbound Rules.
    - Select New Rule and then select Port. Next.
    - Select TCP and enter 1433 for the Specific local ports. Next.
    - Select Allow the Connection. Next.
    - Enter SQL 2014 Instance for the name, for example.

  Repeat the same procedure to add the SQL 2014 Browser (UDP port 1434).

- When moving to SQL2014 or Oracle 12c R1, you must backup the database before upgrading, and then restore and upgrade the database using a server and client on the new platform.

- It is recommended that the Server and Client be on same version of SQL/Oracle.

- It is recommended that only one version of SQL/Oracle is installed on the client and server machines.

- When installing the Name Generator, you must manually configure the Microsoft Distributed Transaction Coordinator (MSDTC) service and add the necessary users to the Distributed COM Users group. See the procedure described in the **Configure Microsoft Distributed Transaction Coordinator** section of the *Smart 3D Installation Guide*.

- Smart 3D recommends client display settings set to 100%. Use of Display Settings larger than 100% may result in distorted menus and text.

- There have been many enhancements to Smart 3D graphics software. We have found that most issues are resolved by loading a current display driver, or rolling back one driver. If the current driver does not fix your particular card's problem, then please copy the "CoreDisplaySettings.ini" file from the following location: *[Smart_3D_Installation_Directory]\Core\Tools\Administrator\Bin* to *[Smart_3D_Installation_Directory]\Core\Shared\Bin* and contact your Smart 3D support representative.
    - Note: If display issues are observed on a client referencing Point Cloud data, Smart 3D hardware should be disabled with the "CoreDisplaySettings.ini" file.

- Smart 3D and Integration with SmartPlant P&ID with Pipe Spec Access: To take advantage of the software's SmartPlant P&ID integration functionality, you must install the Piping Specification Remote Access Server. The remote access server setup is available as an option in the Smart 3D Installation setup for the workstation computer. This feature must be installed on a computer where the Smart 3D client software is installed. The Piping Specification Remote Access Server feature sets up the workstation computer as a remote access server. The Piping Specification Remote Access Client is prerequisite software for the SmartPlant P&ID client whenever connecting to a Smart 3D Piping Specification Remote Access Server. If the Smart 3D Piping Specification Remote Access Server feature is installed, all associated files will be installed and the SP3DPipingSpecRemoteAccessServer COM+ component will be created and registered.

- The Smart 3D Programming Resources component provides developers with the tools necessary to customize the software by creating custom commands or custom programs. Before attempting to create commands or programs, you should be very familiar with Smart 3D interactively and understand its basic concepts of projects, engineering, architecture, concurrency and data stores.

- For best results, format the hard drive before loading any Smart 3D prerequisite and application software and make sure you have enough hard drive space available.

- Microsoft XML Core Services 6.0 is required for extracting Isometric Drawings. Personal Isogen is dependent upon the MSXML 6.0 Runtime Libraries, and if they are not installed on the client machine, Isometric Drawings will fail to update successfully. To determine if this missing requirement is the cause of a failure, check the LOG generated under REPORTS>ProcessFile in the drawing's extraction data for the following text: "Unable to load XML document". MSXML 6.0 is included with Windows 7 operating systems.

- The Name Generator Installation link on the Smart 3D installation dialog allows users to install the Name Generator separately from the Reference Data Installation. For Oracle users, it is recommended to install the 32-bit Name Generator on a system with 32-bit Oracle Client installed. The 64-bit Name Generator can be installed on 64-bit Oracle database server. See the S3D Installation Guide for more details.

- The Name Generator and Piping Specification Remote Access Server installation requires the Distributed Transaction Coordinator service to be enabled.

- When you install Smart 3D Reference Data on the server, it will create a SharedContent (Symbols) folder, but it no longer automatically shares this folder out on the server machine. The share and permissions for the share will have to be manually created. For production use, **it is strongly recommended that you do not keep your model's SharedContent folder under the product folder**. This may cause problems when you need to reinstall Smart 3D Reference Data because any modifications you make in delivered files will be overwritten by the new installation. Instead, you should copy the folder to another location on your server and set permissions that are appropriate for your environment. At a minimum, permissions must allow Read access for all Smart 3D users and Write access to those Smart 3D users responsible for creating Drawing and Report output or responsible for Bulkloading and Catalog management.

- The user logon used to create the Smart 3D databases should be part of the Administrators group on both the Administrator machine and the database server. Other users do NOT have to be part of the Administrators group on the local machine to run Smart 3D.

- For Oracle 12c users:
    - It is recommended that you set up the Windows server network before installing Oracle. Also, Oracle recommends a static IP address. You should install the Oracle server software as the local administrator.
    - The Smart 3D Reference Data Installation delivers three sample script files. These files must be run in order to create the necessary schemas, users and roles used to run the software on Oracle.
        - The Roles scripts have to be executed at least once on each database server. The scripts are located at *\3DRefData\Tools\OracleScriptsToInitDB*. Run SP3DUser_ROLES.SQL before executing SP3DProjectAdministrator_ROLES.SQL.
        - When creating or configuring Administrative Users, the logon used to create the SmartPlant 3D schemas should have "SP3D_PROJECT_ADMINISTRATORS" Oracle role and other settings defined in the *....3DRefData\Tools\OracleScriptsToInitDB\SP3DProjectAdministrator.sql* script. All other SmartPlant 3D users should have "SP3DUSER" Oracle role and other settings defined in the *...3DRefData\Tools\OracleScriptsToInitDB\SP3DUser.sql* script.
    - If you elect to revoke EXECUTE privileges on UTL_FILE and DBMS_RANDOM packages from PUBLIC as Oracle recommends (for security reasons), you need to grant EXECUTE privileges for the "SP3DUSER" Oracle role.
    - When running the Smart 3D Database Wizard, browse for the shared DatabaseTemplates folder that includes the *.DMP files on the server machine, not the administrator machine. This folder needs to have write access for the logged user as well as for the account used as identity by Oracle service.
        - For more information, refer to the *Smart 3D Installation Guide*.

- While modeling in Smart 3D, if you observe that objects are created with "Unspecified" as the name, the COM+ Name Generator service may be stopped or improperly configured. Refer to the *Smart 3D Installation Guide* for the COM+ Name Generator service configuration procedure.

- Supporting internationalization in a homogeneous environment is one of the enhancements available in Smart 3D. A homogeneous environment uses elements from only a single locale. When starting a new project, use extra care during installation and configuration to ensure the proper creation and maintenance of homogeneous environments. Please see the **Internationalization** section of the *Smart 3D Installation Guide* for more information.

- The default security settings for Excel do not allow Trust access to Visual Basic projects. In order to run drawing border and embedded reports from the Drawings and Reports task on computers with Office 2013 (32-bit), you must modify the security settings in Excel to allow Visual Basic projects to run. To change this setting for Office 2013 (32-bit), open Excel and click the File tab. Select Options > Trust Center > Trust Center Settings > Macro Settings and check the "Trust access to the VBA project object model" option.

- IFC service needs to be manually stopped before removing software.

- Smart 3D uses both an internal batch processing system and the Intergraph Batch Services application to help run large, time-consuming operations on a separate computer or during production hours. This leaves the workstation available for other modeling. Due to the intense nature of batch operations, using one or more computers dedicated as batch computers is recommended.

- Smart 3D is compatible with Microsoft Office 2013 32-bit only. Macros are not working when MS Office 64-bit is installed on Windows 7 64-bit and Windows 10 64-bit OS machines. Smart 3D has been tested and certified with Office 2013 (x86).

- In 2016 the *SharedContent/Bin/Layout* folder and files have been removed from the delivered SharedContent directory. If you do not have a copy of these delivered files from 2014 R1 or prior, then they can be found on the SmartSupport Smart 3D content site (https://smartsupport.intergraph.com/).
    - If an upgraded model contains Layout objects based on the Intergraph delivered layout symbols, copy the *SharedContent/Bin/Layout* folder and files from the old SharedContent folder and paste it into your 2016 SharedContent (Symbols) share, so that the *SharedContent/Bin/Layout* folder and files exist in the 2016 SharedContent directory.

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Catalog Data / Reference Data

- When saving the Verification Report output on a Windows 7/8.1/10 machine, a sharing violation may occur and the user may be asked to save a temporary file. This is due to third-party virus protection software. To work around this issue, load the latest service pack or hot fix for Office 2013 and check the online support for your virus software to see if there is any additional information.

- When you bulkload Smart 3D Reference Data workbooks on a workstation with Windows 7/8.1/10, you may be prompted by the software with the Excel "save as" form. The bulkload process is unable to write to the GUID sheet of the workbooks being loaded, so the user will have to save to a new copy of the workbook. In addition, if the original workbooks were in the \Program Files (x86)\ path of the directory structure, Windows may issue a warning stating that you do not have permission to save in the default location. You will then be prompted to save to another folder instead, and Windows will save the new workbook there, as well as save the log file to a directory structure under \[*User Login*]\AppData\Local\VirtualStore.
    - To avoid this:
        - Copy any workbooks being bulkloaded to a directory not in the \Program Files (x86)\ path
        - Run the bulkload process using the copied workbooks and not the original.
- The Bulkload Utility will generate errors if you try to run simultaneous bulkload processes on the same workbooks for a given catalog. If multiple administrators are assigned to modify a catalog database, they need to organize their tasks with this restriction in mind.
- When Cableway specification reference data is being copied from a populated catalog to an empty catalog, the following additional steps must be followed in addition to those described in the Catalog Task Help documentation:
    1. Select the node ...\Cableways\Specifications\Cable Way Specifications\Shapes folder and insert a new row.
    2. Fill in the shapes data and save the changes.
    3. Select the node ...\Cableways\Specifications\Cable Way Specifications node and insert a new row.
    4. Fill in data for the cableway spec in the grid and save the changes.
    5. Expand the node for the new spec and ensure that the shapes node and the bend radius node are created.
    6. Select the node ...\Cableways\Specifications\Cable Way Specifications\ <new spec>\Shapes\<new shape> and insert a new row.
    7. When the new shape row is created, fill in the data and save.

    Once these steps are completed in the empty destination catalog, the user can copy and paste the \Cableways\Specifications node from the populated source destination to the empty catalog. Note that these steps are not necessary when pasting to a populated catalog in which shapes and specs have already been defined.
- Solid Edge requires that the symbols are read/write.
- When bulkloading piping data from the delivered workbooks into a catalog created with the AppRepos.dat template file, you must load both the catalog and specification data workbooks together.
    - The Bulkload Utility needs the settings contained in the DefaultProjectOptions worksheet from the specifications workbook to create the part data for the piping commodities contained in the catalog workbook.

- When using Delete and Replace mode in the Bulkload Utility, all existing catalog entries for each worksheet will be deleted and replaced with the contents of the worksheet being bulkloaded.
    - In cases where worksheets exist in multiple workbooks (i.e. PipingGenericDataBolted, PipingCommodityMaterialControlData, Pipestock etc.), missing data must be bulkloaded in Append mode to restore data that was deleted.
    - When Delete and Replace mode is used with Piping Specification Rule worksheets, only data related to those piping material classes appearing in the bulkloaded worksheet are deleted and replaced. All other piping material classes in the database are unaffected.
- For some cases where copy/paste is not supported for valid reasons, the Spec Editor user interface allows the user to "bypass" the UI by using the CTRL + V key-in when the paste icon or commands are deactivated. Do not use this key-in if the Paste icon is not active for the current node selection.
- The option to skip the Update Object Type Hierarchy and Catalog Views step during Bulkload can result in some time savings if frequent bulkloads are to be performed in a short period of time. However, it is important to perform this update regularly to ensure changes/additions are available to the model. When multiple bulkloads are being performed, the Update Object Type Hierarchy and Catalog Views option should be checked to be performed for the last bulkload at a minimum.
- The location of the Verify Consistency Report may be specified by the user, but the report name will always be Piping Material Class.xls.
- In a Global Workshare environment, bulkloading can only be done at the Host location. It is also expected that the Verify Consistency Report will be run at the Host.
- Users no longer need to enable all macros in Excel in order to run the Verify Consistency Report and Catalog Engineering Check Reports. However, access must still be given to run Visual Basic Projects.
    - To change the settings on computers with Office 2013, open Excel, click the Office button and select Excel Options. Click Trust Center > Trust Center Settings > Macro Settings.
        - Under Developer Macro Settings, check “Trust access to the VBA project object model”.

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Civil

- The start and end invert elevation values of the Trench cross section should be below the elevation values of the 3D Sketch path for the trench in placement methods-1, 2, 3. These values are always with respect to Global coordinate system.
- The user cannot extend a trench run from the end point of the trench run by selecting the Place trench command. The user has to select the existing trench run and modify the 3D Sketch path to continue from the end point of the existing run.
- When placing a branch from a trench run, it cannot be joined to another trench run.

- When placing a branch run, the user should start the trench path of the branch from the 3D sketch path of the header run. Switch off the Point-On Surface SmartSketch option for easy selection of the Header run trench path while placing branch run.

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Common

- When opening Smart 3D, the user is presented with the option to select a session template that best matches their needs for modelling, based on the model type. The user can choose from a Mine template for an MHE model (English Units or metric units), a Plant template for a plant model (English units or metric units), or a Ship template for a ship model (English units or metric units). There is also a General session template that is not unit-specific that can be selected.

- New default colors were added in Smart 3D 2014 R1. To return to the colors in previous versions, open Project Management, select the Model and invoke the Configure Default Colors command from Tools>Project Settings. Select all rows, delete, and save the changes.

- The background color was changed in Smart3D 2014 R1. To return to the colors in previous versions, go to Tools>Options, Colors tab>Background color and set Red-58, Green-58 and Blue-58. To return to the colors in Smart 3D 2014 version set Red-64, Green-64 and Blue-128.

- The default render mode is changed; to revert to the mode in previous versions, go to Format>View and set Render model to “Smooth Shaded”.

- The default light settings are changed. To return it to previous settings, go to Format>View and check the “Use Default settings” at the bottom.

- MSDTC service is no longer required for the client installation. MSDTC must be configured on the Name Generator server.

- Silhouette Edges as a SmartSketch point is turned off by default. In order to use this new feature, it must be turn on in Tools>Options>SmartSketch options. Select the check box called Silhouette Edges.

- Insert File command defaults to the Symbols (SharedContent) share. Users can insert a file both from the Symbols (SharedContent) share and from other locations. A warning message pops up on inserting a file from other locations: “The selected directory is not the symbol share path. In a global work share configuration, you must insert reference files from the symbol share directory if you want other locations to be able to work with these attachments. Are you sure you want to continue?"
    - Yes = Insert the file from the non-symbol share path.
    - No = Return to the Insert File dialog box to either Cancel or browse back to the symbol share

- If you receive application errors, it is highly recommended that you save the session, exit the software and re-enter. In some situations, when receiving an application error, the software may become unstable and may cause data corruption.

- Session files created in previous Smart 3D versions can be used in the same databases after upgrade. The upgraded databases must have the same name as the earlier version and the session files should be refreshed and saved after opening.

- Depending on the graphics card in your machine, you may encounter some display issues. The majority of the display issues can be resolved by simply utilizing one of the View Control commands: Pan, Fit, and Zoom. It is also recommended to update to the latest graphic drivers.

- When hovering over object(s) for a set time, an "Added to List" icon flashes near the cursor to indicate that the object has been added to the SmartSketch® Locate List. This happens while in any command selected in the graphic window.

- Surface style rules will be applied to Parts only, not to Features or Runs.

- Surface Style rules can be applied only to the designed equipment parent and nozzles, not to the shapes or components under it.

- Notes will be inserted only on the Parts, not on Features.

- If runs or features are assigned to Work Breakdown Structure (WBS) objects, defining a workspace with filters using such WBS objects will not display the objects in the graphic view. If you want to see the WBS Projects/Items along with the objects that are claimed/assigned to them, create one filter defined on the WBS tab and another filter defined on the Systems tab. Then, create a compound filter using the “OR” operator to see both the WBS items and the objects selected on the Systems tab.

- For the wildcard character * (*) to work in Filter Properties, the comparison operator must be "Contains" instead of "=". For example, the Pumps "P-1000A" and "P-1000B" exist in the model. To query them from Filter "Properties" tab, select "Match All" and enter a criterion of "Name Contains P*".

- If the filter properties are modified and the session file is not saved before closing, re-opening the session and selecting Refresh Workspace, will not update the graphic view as per the modified filter. The workaround is to select the modified filter once again, define the workspace, and save the session file.

- By default, the Plant Filters folder belongs to the Plant, not to a particular Permission Group. A user must have at least Write permissions at the Plant level to create Plant Filters. If access control is required (e.g. for Global Workshare), it is possible to create subfolders under the Plant Filters folder and assign these folders to specific permission groups and/or locations.

- If an object is brought into the workspace, then all of its associated interferences are automatically displayed even if they are not part of the filter used to define the workspace.

- The same date format as displayed on Object Properties page (date with time) should be utilized in order to filter objects using "Date Created" and "Last Date Modified" attributes.

- When multiple objects are selected, the properties page will display the tabs that are common for the objects selected. However, the Relationships tab will not be shown if multiple objects are selected even though it is part of the common controls.

- To display mnemonic keys permanently on the menus, perform the following steps:
    - On Windows operating systems, go to the Control Panel -> Ease of Access Center -> Make the keyboard easier to use -> Make it easier to use keyboard shortcuts section and check the option to “Underline keyboard shortcut and access keys".

- The Undo command is meant to undo database operations, not local workspace operations. For example, the Undo command cannot undo the orientation of view defined by "View by Points". Use the Previous View (Alt+F5) command instead for this purpose.

- For Equipment and Coordinate Systems, the Common Move command goes directly into the "To Point" smart step by taking the Origin/Control Point as the default "From Point". The user can always go back to the "From Point" Smart Step and redefine it manually.

- If a user locks the vertical angle while in the Pinpoint Spherical Coordinate System, the horizontal angle automatically locks at 0.0. Similarly, if the absolute distance is locked, then the vertical angle is automatically locked at 0.0. However, the software allows locking the horizontal angle by itself.

- When working in an orthographic view and using the Fit command, the object will be fitted to the graphic view with a small amount of space between the edges of the object and the window border. When working in an isometric view, this same Fit process will leave more space between the edges of the object and the window border because of the way the object range is used. If you wish the Fit to be tighter, use the Zoom tool to zoom in closer on the object.

- Paste and Restore is meant for copying from the plant restored for selective recovery into the corresponding active plant. Use Copy/Paste or Model Data Reuse commands instead, for objects that need to be copied from one model to a different model (with catalogs having the same metadata schema).
    - The typical workflow for Paste and Restore is to backup the active plant at regular intervals, Restore the plant from the required backup set using the option “Restore Plant for selective recovery of model objects”, copy from this restored plant and use Paste and Restore in the active plant.
    - Before restoring, ensure that all the objects in the active plant that will be restored are in “Working” status.
    - Before restoring, ensure that user has write permissions to the Permission Groups where the copied objects belong.
    - Currently, branch runs alone cannot be restored. If branch runs are to be restored, copy the header as well as the branch runs.
    - Pasting of features alone may cause problems if there are reducing or transition features adjacent to the features being restored. In such cases, the entire run should be restored.

  - If some objects have a relationship (say between Equipment and a Slab or between Shapes of Designed Equipment) and if some of the objects forming the relationship are deleted from the active plant, restoring only the deleted objects from the backup plant will not restore the relationship. One needs to restore all the objects forming the relationship to restore the relationship as well. Otherwise, these may need to be reset by the user after restoring.
  - Sometimes doing Paste and Restore/Undo more than once for the same set of objects may not place the objects at the correct location. In that case, the objects need to be deleted from active plant and restored from backup plant.

- Users may experience performance problems when using SQL filters especially when using such filters for Surface Style Rules. In such situations, use of these filters should be avoided or limited.

- Software displays a message when E, N, El key-in fields of the PinPoint ribbon bar has an error in the expression or if the name of the plane is wrong. In order to retain the value for further edit in those fields, the user should press the Enter key without moving the mouse point. If mouse point is used to accept the message or the mouse hovers in the graphic window after pressing Enter key, the software will clear the entry in the field. In such situations, the user needs to enter the complete expression again. Tip: Use mnemonic keys instead of the mouse to bring the focus back to the pinpoint key-in field (For example, use Alt+N for getting to N field).

- By default, Copy/Paste of objects will result in objects pasted in place. If the user wishes to move objects dynamically while pasting, he/she can uncheck the “Paste in place” checkbox on the Paste Special dialog. When the “Paste in place” option is unchecked, Paste allows the user to dynamically select the placement point.

- If the filter used to define workspace returns the expected objects but no objects are seen in the graphic view after using the Fit command, then the range of objects in the Workspace Explorer most likely exceeded the precision limit mentioned previously. This could be caused by objects from externally referenced files or Reference 3D Models positioned far from S3D objects. Users should either position these external references closer to the S3D objects or use volumetric filters. Volumetric filters should be used if the references are properly positioned but are so far away from SP3D objects that a filter with a combination of S3D objects and referenced objects exceed the supported range.

- If the user selects "Referenced File" on the Object Type tab when defining a filter and does not select any item in the Reference tab, then the query will not return anything. The user must select both the Object Type and the actual referenced file.

- When a user inserts a "Referenced File", he/she will not see any graphics in Smart3D unless the workspace is defined with all References selected using the “Include nested objects” option. If the workspace is defined in this way and a reference file is not seen, the user should redefine the workspace filter to include the inserted file desired.

- Inserted MicroStation and AutoCAD files can be moved with the SmartPlant 3D Move command. However, only the coordinate triad of the file should be moved. The graphics will not dynamically move with the triad. Once the triad is moved, the user must refresh the workspace to view the graphics in their new location. Inserted graphics cannot be moved independently of the coordinate triad. The recommended procedure to move inserted graphics is to select the file name from the Reference Tab of the Workspace Explorer window. This will highlight the coordinate triad in the 3D window. Then, select the Move command and complete the move steps as desired.

- When creating Volume Filters for use with external (non-SP3D) objects such as "Referenced File" or “Reference 3D Model”, users should expand the filter definition to include the "Referenced File" or “Reference 3D model(s)” on the respective filter tab.

- Users can insert 3D AutoCAD files for both .dwg and .dxf formats. Some AutoCAD files are “unit-less” and SmartPlant 3D will not know how to scale them to match the 3D model. To account for this, there are settings that can be modified in the Translator.ini file. This file is located in [*Smart_3D_Installation_Directory*]*Core**Runtime**SPFSPRB*. For more information, contact Intergraph PPM Support.

- As not all object types are supported when inserted, examine the log files generated during the insertion process to see if some graphics were not imported properly. For example, the import of .dgn files does not support SmartSolids & SmartSurfaces (Type 2), SmartGeometry (Type 38) and nested Shared cell definitions, which is a shared cell instance (Type 35) under shared cell definition (Type 34). To examine files before insertion, the EdG command, “set search/type=2,34,35,38”, can be used to see what needs to be modified in order for the object type to be visible in SmartPlant 3D.

- To protect Style Definitions within the model, users should assign Styles to a permissions group that has controlled access. To do so, first clear the select set in the workspace (by pressing the “Select” arrow in the vertical toolbar) and open the Format Style dialog. Modify the “Permission Group for Styles” and press Apply.

- Ship-related cases have a variety of Data Reuse (Copy by Family) functionalities.
    - Main functions:
        - **Copy Similar Family** - Creates a family using which objects can be copied from source to target within the same model.
        - **Copy Symmetry Family** - Creates a family using which objects can be mirrored across L0 from source to target within the same model.
        - **Copy Across Models Family** - Creates a family using which objects can be copied from source to target in different models pointing to the same site & catalog.
    - Copy by Family operations:
        - **Find & Correlate Command** - Used to find the identical structure from source to target to avoid duplicate copies on the target.
        - **Create Target Copy Command** - Used to copy the structure from source to target.
        - **Modify Target Copy Command** - Used to propagate design modifications from source to target.
        - **Delete Target Copy Command** - Used to propagate object deletions from source to target.

- The Copy by Family command can be executed from Molded Form/Detailing tasks (Edit -> Copy by Family).
- The Copy by Family command now supports design & standard members.
- The Copy Across Model command now supports structural detailing and manufacturing.
- In Sketch 2D, dimensional constraints are maintained when Add Item for reference is used.
- For closed sketches recognition the "Maintain relationships" feature needs to be turned on.

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Common Route

- By default, Fast-Mode Routing is set to "On" (meaning you will not see Turn Features while in dynamics). To turn Fast-Mode Routing Off, press Shift + F before selecting a Route command.
- To achieve the best results while routing using Cardinal Points, always route using the Routing Planes.
- To easily select an End Feature without selecting the surrounding Cardinal Points, turn off the Key point option on the SmartSketch tab in the Tools>Options dialog. Turning off the Key point option is a best practice if the user wishes to paste a pipe run to the end feature of another pipe run. Otherwise, the pasted pipe might be skewed. However, leaving the Key point option selected is also very helpful in other workflows. For example, to move a pipe surface a certain distance away from the top of steel, use the Move command and select the Bottom of Pipe Cardinal Point, which is the bottom pipe surface, as the “Move From” point.
- It is recommended to turn off the “Point on Surface” SmartSketch glyph when routing pipe out of an Equipment Nozzle in close proximity to a Slab face. For example, when the “Point on Surface” glyph is turned on, it can be difficult for the user to route up out of a Discharge Nozzle on a Pump because the face of the Slab is in the SmartSketch list, which causes the software to lock to the elevation of the Slab face instead of allowing the user to route the pipe to a higher elevation. The “Point on Surface” glyph can be turned off on using the F3 key or the SmartSketch Options toolbar.
- After selecting the route starting point, the user can open the properties page and on the defaults tab, can change the Remember Working Plane to True. This is a session setting that holds true across all routing disciplines.

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Compartmentation

- Starting with version 2016, Compartmentation supports Model Data Reuse, Model Data Transform, and delete optional functionality.

- Attribute computation during compartment creation or modification may cause longer processing times. The timing of when a compartment's rules are executed is controlled by the *SemanticControl* property in the *CmpartCustomAttribute* sheet of the *CompartGenericServices.xls* workbook.

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SmartSketch Drawing Editor

- The SmartSketch Drawing Editor can now be invoked directly from the Intergraph Smart 3D entry on the Start menu. This is beneficial for users that want to edit standalone drawings.

- Previously, the Drawing Editor, which is now called the SmartSketch Drawing Editor, could be started in a standalone mode without pulling a Smart 3D license from SmartPlant License Manager (SPLM). The new policy requires that the SmartSketch Drawing Editor pull a license when it is invoked in standalone mode. This applies when the user either double clicks on a *.sha file or starts the product directly from the Start menu. This will not affect the usage of the SmartSketch Drawing Editor from within the normal Smart 3D tasks.

- It is possible to define transparent line styles within the SmartSketch Drawing Editor. Such objects will not be visible when viewed with the Drawing Editor. However, when files with these styles are exported to AutoCAD or MicroStation, the graphics become visible since those products do not support transparency. A new setting called Export Transparency Threshold is available to avoid this problem. This is an integer between 0 and 100. During translation, objects are inspected and if their transparency is over the value in the configuration file they are not exported. The default value is 100, meaning objects are translated regardless of their transparency. Setting Export Transparency Threshold = 99 will prevent objects which are 100% transparent from being exported. This can be added to the Options section of the translator configuration file, which is itacad.ini and itmstn.ini located in *<Smart_3D_Installation_Directory>\Common2D\Rad2d\Bin*.

- It is possible to merge the 'multiple files' that gets created when converting a SmartSketch Drawing Editor file (*.sha) to AutoCAD or MicroStation. The setting 'Merge References = 1' needs to be added to the Options sections of the translator configuration file, which is itacad.ini and itmstn.ini located in *<Smart_3D_Installation_Directory>\Common2D\Rad2d\Bin*.

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Drawings

- When updating large drawings in 64-bit, you may experience a performance impact if the batch server has to rely too heavily on virtual memory. This impact can be reduced by adding more physical memory to the batch server. To determine if the system has reached the point of consuming virtual memory, monitor the Drawing Sheet logs and the SP3D errors logs. If the memory requirements exceed the physical memory of the computer, the update performance will be impacted. For example:

  Example from SP3DErrors.log

  *** Info ***

  Time : 07/09/13 03:29:23

OS : pid,3884, elapsed,29311.432, cputime,6264.439,secs, memory (commit,5684.305, reserved,25939.988, code,434.938, stack,0.215,) MB
ID : 0x00000000(0)(0);DRAWINGS
Source : Update::Execute
Desc : Finished Updating Drawing: \OP7108_ModelUpgrade\Alphas\Composed\ISO VIEW
Help : Update.cpp;518

At the time the message was written to the log, the process was consuming 5684.305 + 25939.988 + 434.938 = 32059 megabytes of memory. Divide by 1024 to convert megabytes to gigabytes gives ~31 GB. If the batch server has less physical memory than that, then virtual memory was used for the update.

Similarly, from the Drawing Sheet's Error log (right-click menu -> View Log):
[Information - Reported on: 07/08/13 at 22:56:43]

Process Virtual Memory: 31193 MB
Extra Information: Begin RadGen. This includes Label and Dimension point generation.

- Users who will be creating or modifying any portion of the Drawings Catalog (for example: view styles, borders, packages, keyplans, etc.) must have at least Write permission on the Symbols (SharedContent) share.

- Composed, Volume and Orthographic Drawings by Query drawing types can contain more than one drawing view. Each view can have its own orientation, scale and view style. For Composed drawing types, each view is represented by the same or different volume in the model. For Volume drawing types, each view in a drawing refers to the same drawing volume in the model. For Orthographic Drawings by Query drawing, each view is represented by a single object/assembly/system around which an imaginary volume is created during the drawing extraction.

- If **no** Graphic Rule is specified in a particular view style, then all objects included in the view style will have a visible line style of Normal, and hidden lines will not be displayed.

- When using the Place Drawing Volume by View command in Space Management, the user can change the location of the cursor on the volume face (cardinal point) by using the left and right arrow keyboard keys while still in dynamics. A movement of the mouse is required to see the new location.

- Do not use filters from the My Filters folder in view styles.

- The Drawings by Query method to produce drawings does not support the use of My Filters.

- If an out-of-date drawing has both property changes and graphic changes to be incorporated into it, updating the drawing using the Batch commands will only incorporate the property changes into the drawing. A second update is required to incorporate the graphic changes. The Update Now command will incorporate both property and graphic changes into the drawing with one execution of the command.

- When exporting to 3D MicroStation files, it is recommended to turn clipping off in the view styles to improve the fidelity of the results.

- When specifying an output path for 3D Model Data or DGN components, it is a good idea to use a UNC file path. If the component is updated from a workstation different from the one that it was created on or it was updated via batch, a hard-coded path may not generate results to the intended location or errors could occur. This same concept applies to batch jobs that are submitted to other servers to perform updates. Output will fail to be saved on the server unless UNC paths are used.

- For Drawings to contain embedded reports it is necessary to have the Microsoft Excel “Trust access to Visual Basic Project” option enabled. This is a ‘per user’ setting, so it must be established for all users who will embed reports or update drawings with reports. If a drawing report fails to update, check this setting to confirm it is correct. Also, refer to SP3DReports.log for more details. Even if the user embedding the report has the correct settings, the report will still not update if the user updating the drawing does not also have the correct settings. For a batch server, this must be done by the user who sets up the batch processing.

    - From the Microsoft Office Button, select Excel Options button, then Trust Center category, then Trust Center Settings button, then the Macro Settings category. You should then check the box adjacent to "Trust access to the VBA project object model".

- A licensed copy of MicroStation J and/or MicroStation V8 is required to be installed on workstations updating 3D DGN documents. This also applies to workstations that are configured as batch servers if the 3D DGN documents are to be updated in batch. It is possible to install both versions of MicroStation on the same machine.

- Drawings can include embedded reports. The reports can be formatted in an Excel document or graphical textboxes. There are Microsoft limitations that restrict the number of rows and columns that an Excel spreadsheet can display when embedded in another document. It is not a fixed number of columns or rows, and a combination of factors determines this limit. In general, the limit is low. There is no indication to the user when this limit has been exceeded. Therefore, users should be aware that embedded Excel reports may be incomplete. It is suggested that embedded reports be placed using the textbox option, for which there is no such limitations.

- Functionality exists that allows users to turn off textbox borders when embedding reports in drawings, which is relevant only when a user has opted to insert the report as textboxes instead of using Excel. In this case, the Excel cell border style property will be used to determine the weight of the textbox border. If no explicit border is defined for a cell, a border will not be placed around the corresponding text box. This has little, if any, impact on delivered reports. Users may find it necessary to modify their Excel layout templates for custom catalog reports. For all data rows to have visible borders, the report must be defined with the “Insert Values, Copy Formats and Formulas” Filling Mode option selected. It may also be necessary to add or remove cell borders to get the desired output results when converting embedded reports to text boxes.

- Since BSpline Surfaces in MicroStation design files are sometimes hard to process by third party applications and also consume a great deal of memory, Smart 3D Export to DGN exports most of the Smart 3D GTypes (such as GRevolution, GRuled, GSphere, GPlane, GTorus and GCone) as MicroStation Solid/Surface/ComplexShape/Ellipses/Cones.

- To create “L-Shaped” volumes for use with composed drawings use the place volume along path command to create the volume. Using merged volumes or volumes bounded by spaces will not work for this purpose.

- To improve performance there is a new requirement that package.xml files must reside in a folder that matches the name of the package ID. All delivered packages meet this criterion. All packages created using the Save Package command will meet this criterion. If a package has been added manually then it may not meet the criteria. In this situation, a message will be displayed in the Drawings.log file stating “Package <name> not found in SymbolShare”. Changing the name of the folder to match the package ID will correct the problem.

- A batch timeout feature has been added to batch updates. Currently no error or information log is generated if a timeout does occur. This information can be found in the Event Viewer. The timeout information will also be sent in an email if the user has specified an email address when submitting the batch job. It is not necessary to check any of the Job Start, Job Completion, or Job Abort options when submitting the job, the email address alone is sufficient.

- If the shared content is upgraded from previous versions (i.e. V2011 R1 or V2014 R1), then the view styles require an upgrade using the Migrate Dimension Rules.exe located under *<Smart_3D_Installation_Directory>\Core\Container\Assemblies\Release* folder.

- Some commands on the 2D Drawing Editor toolbar are not available when opening a document from the Drawings & Reports task:
  - Place Snapshot View
  - Associate Objects to View
  - Remove Associated Inputs
  - 2D/3D Selection

- A Drawings tab has been added to the Workspace Explorer which can be used from any graphical task.

- “Full Update” can be used for drawings by rule on any level in the hierarchy of the drawings documents. It will only run a full update on drawings by rule components.

- ‘Ctrl + C’ and ‘Ctrl + Right Mouse Button’ no longer can be used to copy views. Only the “Copy View” command can be used to copy a view.

- If you are registered with SmartPlant Foundation, the Publish and Revise commands are added to the drawing menus.

- If batch processing has been configured (by an administrator), the Batch command is added to the drawing menus.

- Drawings are not dependent on the defined workspace.
  - Any object in the model can be drawn, even if it is not currently loaded in the workspace
  - Query results therefore may contain objects that are not in the current workspace

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Electrical

- Modifications of Cableways containing a very large number of Cables (500+) may take a long time. This is because each Cable has relationships to the Cable Tray features that must be updated. Delay cable update command can be invoked in such scenario (Shortcut: ctrl +Shift + C in Electrical task) where the user can defer the processing of cables when cableways are modified. To Do List entries are created for the cables that can be resolved later.

- During the Edit Cable Path command's Set Entry and Set Exit steps, the following items can be selected:
  - Any point on the Cable Tray/Cableway Straight Feature (including the End Feature)
  - Cable Tray/Cableway Turn Features and Along-Leg Features
  - Conduit End Features
  - Hanger or Support on Conduit Run or Cableway

- The Edit Cable Path command does not "jump gaps" when routing a Cable. That is, if two disconnected Cableways are selected, the software will not automatically connect them with a path leg (same object used to connect Entry/Exit point to equipment). In this case, the cable path will have a status of "Incomplete", and the graphics will appear disconnected. To correct this, connect the two non-contiguous Cableways with a third Cableway (use the "zero spec" if no parts are desired).

- There are two modes for editing a Cable Path: (1) the "Edit Cable Path" command button on the vertical toolbar and (2) the "Select" command. Either one can be used with similar results. If a Cable is selected via the "Select" command, the ribbon bar will display the group of SmartSteps making up the Edit Cable Path command. Selecting any one of these buttons (Select Cable, Set Entry Point, Select Way Features, Select Avoidance features and Set Exit Point) can be clicked to start routing the Cable. Selected Cables(s) can also be routed without selecting the Edit Cable Path command on the vertical toolbar.

- When routing Cables, if more than one is selected and one of the Edit Cable Path SmartSteps is deployed, the software compares the paths of all of the selected Cables with that of the first Cable selected. If they are different, a message displays notifying the user that he has two choices: (1) All paths that are different will be adjusted such that they match that of the first Cable**. (2) All Cables having different paths will be dropped from the selection. This is done so that the software will not have to maintain heterogeneous Cable Paths during the routing process. Note that no changes are saved to the database until the FINISH button is selected.

  - NOTE:

    This functionality can be used to copy one Cable's Path to one or more other Cables:

    8. Select a Cable with the path to be copied.
    9. Select other Cables that will have the path copied to them.
    10. Pick the Select Cable(s) button or one of the other SmartSteps on the ribbon bar as if you are going to route the Cables.
    11. When the message appears, select "OK" and the Cable Path will be copied to the other Cables.

12. Select the FINISH Smart Step to save the paths to the Plant database.

- A Cable Tray component can be inserted by Port 3 using the Flip feature on the ribbon bar. Users can select the Port for placement and use the Reference Position dropdown to select the Port's orientation on the feature. Port position cannot be altered after placement.

- The size and shape for a selected set of Cableway features can be edited on the Cross-Section tab of the Property Page.

- Before Auto-routing Cable, the user needs to set options in the Auto-routing options dialog. For example, if the user would like to auto-route a Cable entering a Cable Tray straight feature, the "Allow cables to enter from the cableway straight feature" check box needs to be selected. Review the other options on this dialog.

- Range boxes for modified Maximum Tail Range values are updated only on reselection of Cable.

- To facilitate Auto-routing of Cable, you need to select the Signal Type for the Cable Tray before auto-routing Cable.

- Users can fence select to find Along Leg Features while in the "Select Way Features" SmartStep.

- After Auto-routing, please review the status in CableAutoRoute.log in the %temp% folder.

- Fence select to add or remove Conduit end features and Cableway end features from your select set when branching out of Duct Bank or to continue routing Duct Bank.

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Equipment and Furnishings

- Users should not insert design files with more than one Equipment as imported Equipment.

- The Place Shape command displays the Shape types on a shape palette for quick selection. To activate the shape palette, click and hold the Place Shape command button on the vertical toolbar for about 1 second. The palette can be moved, resized and left open while using other Equipment commands.

- Use the F3 function key as a shortcut to the "Point on Surface" option on the SmartSketch Tab of the Tools > Options dialog box. The F3 function key toggles the "Point on Surface" SmartSketch option On and Off. Toggle the "Point on Surface" option Off to allow the selection of Control Points behind surfaces for Connect relationships. Toggle On when creating Mate, Align, etc. relationships to allow the selection of surfaces. However, please note that it is recommended to turn off the "Point on Surface" SmartSketch glyph when routing pipe out of an Equipment Nozzle in close proximity to a Slab face. For example, when the "Point on Surface" glyph is turned on, it can be difficult for the user to route up out of a Discharge Nozzle on a Pump because the face of the Slab is in the SmartSketch list, which causes the software to lock to the elevation of the Slab face instead of allowing the user to route the pipe to a higher elevation. The "Point on Surface" glyph can be turned off on using the F3 key or the SmartSketch Options toolbar.

- Keyboard arrow keys can rotate an Equipment, Equipment Component, Shape, and/or Nozzle during placement while in transient/dynamic mode or during modification in the Modify command. The Right and Left arrow keys will rotate the object 90 degrees. The Up-arrow key will change the axis of rotation. If the keyboard focus is in the commands Offset field, then simply click the Graphic View's title bar to activate the arrow keys for rotation.

- Hiding an Equipment Component and/or Shape does not hide the object graphically. To hide an Equipment Component or Shape, hide the parent Equipment. When objects are hidden, they are italicized in the Workspace Explorer and are not selectable in the graphic view. To display an Equipment Component and/or Shape, select the parent Equipment in the Workspace Explorer and select Show. Show and/or Show All can also be applied from the Tools menu.

- Pay close attention to which object, Equipment, Equipment Component or Shape, is selected when creating relationships. Equipment Component relationships can only be created with other Equipment Components (and their children shapes) within the Equipment Component's parent Equipment. Shape relationships can only be created with other Shapes within the Shape's parent Equipment, parent Equipment Component or parent Solid. Create relationships with the parent Equipment object to constrain all Components, Shapes and Nozzles within the Equipment.

- To rotate an individual Equipment Component or Shape with the Rotate Object command, make sure the individual Equipment Component or Shape is selected and not the parent Equipment.

- All relationships/constraints, display aspects and properties of a particular Shape can be modified after placement.

- A Shape's Type (RTCircularCylinder 001, Platform1 001, etc.) cannot be changed after it has been placed. If the type is undesirable, delete the Shape and place the correct type.

- A Nozzle's Type (Piping Straight Nozzle, HVAC Port, etc.) cannot be changed after it has been placed. If the type is undesirable, delete the Nozzle and place the correct port type. All location parameters and properties of the particular port type can be modified after placement.

- If an Equipment's Reference Coordinate System (selected from the right-click menu of the Equipment or Equipment Component) has been deleted, modify the Equipment or one of its Equipment Components or Shape's position, Rotation or Occurrence properties, and the Coordinate System will be regenerated at its original location.

- An Equipment's Reference Coordinate System (selected from the right- click menu of the Equipment or Equipment Component) can be moved, rotated, etc. without modifying (move, rotate, etc.) any one of the Equipment's Components, Shapes or Nozzles. This includes only Nozzles that were not placed with the Equipment as its reference. All Nozzles placed with the equipment as its reference will modify (move, rotate, etc.) with the Equipment's Reference Coordinate System.

- Status or permission group modifications will propagate to Nozzles, Shapes, Equipment Components and Equipment. For example, changing the status or permission group of a Nozzle will propagate up to the parent Equipment and all of its Equipment Components, Shapes and Nozzles. Likewise, if the status or permission group is changed for Equipment the modification will be propagated down to each Equipment Component, Shape, and Nozzle within that parent Equipment.

- The Replace Equipment command does not maintain any Occurrence attributes, Equipment Relationships, Shapes or Nozzles of the original. The entire Equipment or Equipment Component will be replaced with a new Equipment or Equipment Component with default properties. If the Shapes and/or Nozzles of the original Equipment are needed, then copy the objects to catalog and paste them to the new Equipment.

- The Material Name and Material Grade must be set for all Solids that are a child of a Designed Equipment or a Designed Equipment Component in order to view the resulting Surface Area, Volume, Weight and Center of Gravity values for the Designed Equipment or Designed Equipment Component. The Material Name and Material Grade must also be set on the Designed Equipment or Designed Equipment Component itself.

- If a Designed Equipment contains Designed Solids using both the Detail physical and Simple physical aspects, then the Weight and Center of Gravity values for the Designed Equipment will be the sum of its Detail Physical Solids only. If the Designed Equipment contains no Detail Physical Solids but does contain Simple Physical Solids, then the Simple Physical Weight and Center of Gravity values are summed up for the Designed Equipment.

- When inserting a .SAT file using the Place Imported Shape from File command, it may not place at the desired size. The graphics in the file can be scaled by editing the third line in the .SAT file, which looks something like the following:

  1000 9.9999999999999995e-007 1e-010

  The first number is a reference to the Units. In this case, it is 1000. For example, if this value is changed to 100, then the file will insert at one tenth the original size.
  Note that .SAT files are in ASCII format and can be edited with a text editor. However, be careful not to modify other values as this could corrupt the file.

- When the 'WCG Origin' value for Equipment or Designed Equipment is changed to 'Compute', then its Weight is calculated by summing the total Weight of its children objects. The children objects may either be Designed Solids or Equipment Components. If any child object's Weight is undefined, then the parent Equipment's Weight becomes undefined as well. Similarly, an Equipment Component's Weight is calculated by summing the total Weight of its children objects.

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Geometry Analysis and Repair

- Geometry Analysis and Repair (GAR) is unique because objects are not saved to the database.

- The Refresh command will remove everything in the session, and it is not possible to define a workspace with existing model geometry.

- Any changes must be saved to an *.igs or *.sat file before exiting the GAR task.

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Grids

- Due to the 32-bit precision limitations of graphics cards, objects in the model should be within 10 kilometers of the Global Coordinate System, or they will not display correctly. If your plant coordinate values are larger that this limit (for example, E = 200,000, N = 300,000), in order to get the coordinate readout desired, you should define a coordinate system at correspondingly large negative values (in our example, E = -200,000, N = -300,000). You should then use the coordinate system you created as your active coordinate system for modeling and output. Do not bring this new coordinate system into your workspace, or you will have the display problem you want to avoid.

- The Grid wizard is useful for creating a rectangular grid, a radial grid or a combination of the two. The user must select the type (Rectangular, Cylindrical or Both) on the first page of the wizard.

- A set of grid lines on the same elevation plane can be hidden simultaneously by selecting the elevation plane that the grid lines are displayed on, and applying Tools > Hide. A single grid line can also be hidden simply by selecting the grid line and applying Tools > Hide.

- Grid lines can be removed or hidden from view for a particular elevation plane by using the Elevation Plane Position feature on the grid plane ribbon bar and removing the desired elevation plane. The display can be turned back on by using the same feature to add the previously removed elevation plane.

- Grid Planes can have slope and be angled about a rotation axis using an Elevation Plane, a user defined distance, or a combination of both as reference. Elevation Planes cannot be angled about a rotation axis.

- Grid Systems can be exported and imported using both XML and Excel formats. The user should be sure there are no coordinate systems with the same name in the model before using the export command. The export and import commands can be found under File>Export/Import.

- The Grid Wizard proves to be a convenient tool in placing a coordinate system if the user prefers to enter orientation information for the coordinate system into the software, rather than placing a coordinate system by three points. If the user does not want any grid or elevation planes associated with the coordinate system, those steps may be skipped during the Grid Wizard command.

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Hangers and Supports

- Delivered Hangers and Supports parts and assemblies have been converted from VB6 to .NET.

- Anvil Parts have been converted to .NET.

- User Defined Forms are now available for Supports. The AISC LRFD-3.1 W Section Part and the S3D Support U Shaped Frame L Section Assembly display the User Defined Form by default when placing.
- New libraries delivered in Additional Data Files are Oglaend Parts, Seonghwa Parts and Wookwang Parts.

- An option is provided for users who wish to retain the Support when the related route feature is deleted, with a To Do List record being created. The control over deletion of Support when the related route is deleted is given to the user by a Catalog Rule. A new "HgrSupDeleteSupportsWithRoute" rule is added to the "HgrRules" sheet [*HS_System.xls*] and its type is Integer. By Default, the value is set to '1' (meaning true). User can change this value to '0' [Zero] to stop deletion of support when its related Route is deleted. If set to '0', the support goes into To Do List instead of getting deleted. If the rule is absent in the catalog, or if the value is anything other than '0', the current behavior would be continued (i.e., the Support will be deleted).

- During and after placement of a Support Part, the arrow keys are enabled to rotate the object. There is a temporary Coordinate System at the cursor location on the active port and the port orientation is shown in Yellow color. The active axis of rotation is shown in Blue color.

- A control is available on the Place Part Ribbon Bar which allows the user to select a Cardinal Point by which to place secondary steel (Rich Steel). This is similar to the Cardinal Point control present when placing a Member System from the Structure task.

- It is possible to place any Support without selecting a supporting item with the "Place Support by Reference" command available on the vertical ribbon bar.

- The "Toggle Connection to Feature" button on the horizontal ribbon bar has been renamed to "Change Support Configuration."

- Several Locate Filters from the Structural Task are available in the Hangers and Supports Task. These include Member Parts, Member Systems, Frame Connections, Member Assembly Connections and Assembly Components. The Trim Member command from the Structure task is available along with the Place Linear Member and Place Assembly Connection commands in the Hangers and Supports Task. Using these commands, you can quickly add Structural Members and connections to Supports without switching to the Structure task. The Structure Member Part Modify ribbon bar will display with the selection of a Member Part, allowing the properties to be modified as in the Structure Task. Member Assembly Connections and Frame Connections can also be modified in the Hangers and Supports Task.

- Drawings Packages available include the HngSup3View and HngSup3View Style2. These packages provide labeling for various entities as in the HngSup3View drawing package. These include Key plan and Key Plan Callout, Notes, Elevation Callouts, CAD Details, ordinate dimensions and Welds. The sample Support Assembly, Assy_FR_IT_LS, is delivered with added Control Points for the Ad Hoc Note, CAD Detail, Elevation Callout and Key Plan Callout. These are included with respect to the labeling in the HngSup3View Support drawings. A sample Support, Assy_FR_IT_LS_Weld, is delivered with an added Weld Component, with respect to the Weld label.

- Enhancements have been made to the 3rd Party Application which provides an interface to the Lisega LICAD Program and other similar ones. It allows the designer to design a Support, and then automatically instance all the parts into the Smart 3D model. Additions include the offset of parts in the East and North directions and the rotation of parts about the East/West and North/South axes. The Help Tab is updated with the new information.

- New parts have been added to the SmartParts catalog delivered in *[Smart_3D_Installation_Directory]\CatalogData\Bulkload\DataFiles* folder: HS_S3DParts.xls and HS_S3DParts_Codelists.xls.

    - **IMPORTANT**: For migrated catalogs, prior to loading HS_S3DParts.xls, the user must bulkload HS_S3DParts_Codelist.xls and new worksheets and codelist entries in HS_System_Codelist.xls.

- Two Weld components, which can be placed as Parts, are delivered with the SmartParts catalog. They can be found in SmartParts>S3D Standard>Miscellaneous>Welds>Point (S3Dhs_WeldPoint-1) and Rectangle (S3Dhs_WeldRect-1)

- Using the command, Drop Standard, you can convert a standard Support to a Design Support with parts. With Drop Standard, the Name Rule of the Support will be retained.

- The Place Part command can be used in conjunction with Design Supports to create unique Supports for which there is no standard. It can also be used to modify a standard assembly Support to work in special circumstances.

- Design Supports can be placed by enabling the “Design” check box on the right end of the ribbon bar when starting the Place command (by Structure or by Point). The box is checked before committing the selection of the feature. Likewise, if a Standard Assembly is desired, the “Design” box is un-checked before committing the selection of the feature. The “Design” box can be checked or unchecked any time the Straight Feature Smart Step is active, even when returning to it after committing the feature selection with the green check.

- When a Support is copied and pasted, the user has the option on the Paste dialog box to uncheck “Paste in place” allowing the Support to be moved to the desired location while pasting. The user also has the option to change the Structural Geometry or the Straight Feature from the Paste dialog, or to select Delete Optional, which will allow Supports to be pasted without connections. If moved to a valid location, Supports will now Auto-Connect to the new Feature and Structure, or it can be pasted in free space for later connection to a Feature and Structure. In the latter case, the Support will go to the To Do List. When using the Copy command, a “From” reference point must first be selected in order to continue with the Copy process.

- The Mirror Copy command from the Edit pull down menu can be used for Assy_FR_LS_LS and Assy_RR_DR_WS assemblies. Other Supports will be copied but not mirrored.

- You must define the Min/MaxDistance attribute of all the Support definitions in the reference data using the same unit type. They should be defined all in 'in' (inches) or all in 'mm' (millimeters), but not a mix of the two. If the units are mixed, the Type field will be blank and the Assembly Rule will not be used when the Rule box is checked.

- The Hangers and Supports task includes a parts library named HS_Util_Metric that contains a subset of the parts in HS_Utility library. The HS_Utility library parts are documented in the *Hangers and Supports 3D Symbols Reference Guide*. This guide also includes the HS_Util_Metric parts.

- Most of the delivered Hangers and Supports workbooks contain an “Instructions” sheet with information regarding prerequisites, required .xls files, load order and mode for bulkloading into a catalog.

- Libraries include BergenPowerNF09.3 Assemblies and Parts, S3D Supports Steel Frames and Struts, S3D Standard Parts, Anvil PH-2010 Parts and Hilti Assemblies and Parts, delivered in *[Smart_3D_Installation_Directory]\CatalogData\Bulkload\DataFiles* folder.

- Some libraries delivered in *[Smart_3D_Installation_Directory]\CatalogData\BulkLoad\AdditionalDataFiles* folder include FINL Assemblies and Parts, FLSample Assemblies and Parts, S3DParts_JIMC-Common, S3DParts_JIMC-Metric, S3DParts_JIMC-Unified and S3DParts_PTP and 3DAPI Sample Assembly and Parts (.NET).

- HS_Bline_Tray, HS_Halfen_PC, and HS_Lisega component libraries for placing parts are provided with the Additional Data Files. Also provided are HS_Bline_Assy and HS_Halfen_Assy assembly libraries.

- A Ship Style Generic Support Library is delivered that includes a Penetration Plate. The workbooks are delivered in *[Smart_3D_Installation_Directory]\CatalogData\BulkLoad\AdditionalDataFiles* folder. These include HS_GenericAssy.xls, HS_GenericParts.xls (and Codelists), HS_ShipSteel.xls and HS_Correspondence.xls workbooks. When bulkloading the files for the Ship Style Generic Supports, it is also necessary to bulkload the HS_Util_Metric workbooks.

- The Pipe Support Limited (PSL) and Power1 libraries are delivered in *[Smart_3D_Installation_Directory]\CatalogData\BulkLoad\AdditionalDataFiles*. These include HS_PSL, HS_PSL_Codelist, HS_Power1_Assy, HS_Power1_Parts and HS_Power1_Assy_Codelist. The Power1 library consists of additional Pipe Supports based on the Anvil catalog.

- A set of Generic Standard Supports is provided, which consists of “Symbolic Restraint Supports” with the proper restraint information provided. When this information is defined in the PCF file, pipe stress analysis programs, such as CAESAR, will be able to read this information directly. These Supports have simple symbolic graphics, and once the stress analysis is completed, they can be replaced with a physical Standard Support of the same type. The HS_RestraintSupports.xls workbook is delivered in *[Smart_3D_Installation_Directory]\CatalogData\BulkLoad\AdditionalDataFiles* folder.

- When placing Supports on in-line components, a default Variable Cylinder Generic Support is presented in the “By Rule” dropdown list. For other Supports to appear in the "By Rule" dropdown, it is necessary to bulkload the added Supports (in Add, Modify or Delete mode) with the SupportedFamily property set to "PART" or "END" in HS_Assembly.xls.

- HVAC Support Assembly and Part catalogs are delivered for rectangular and round Duct in *[Smart_3D_Installation_Directory]\CatalogData\BulkLoad\AdditionalDataFiles* folder.

- Cableway Supports are included in the TrayShip library, providing Support assemblies for Cable Tray Straights, Turns and Tees, as well as the Angle Bar Supports. The workbooks are delivered in *[Smart_3D_Installation_Directory]\CatalogData\BulkLoad\AdditionalDataFiles* folder.

- A library of Marine Supports is delivered. The workbooks are delivered in *[Smart_3D_Installation_Directory]**CatalogData**BulkLoad**AdditionalDataFiles* folder. The necessary files should be bulkloaded as instructed on the “Instructions” sheet in the workbooks.

- A library of Rich Steel is delivered that provides the ability to automatically trim HgrBeams. The Cutback mode is supported for all Cross Sections except Circular, and the Snip mode is supported for the L Section. A set of steel files based on the Rich Steel is delivered and named to match the structural workbooks upon which they are dependent.
  - Delivered in [*Smart_3D_Installation_Directory]**CatalogData**Bulkload**DataFiles* folder:
    - HS_Str-AISC-LRFD-3.1.xls
  - Delivered in [*Smart_3D_Installation_Directory]**CatalogData**BulkLoad**AdditionalDataFiles* folder:
    - HS_Str-AISC-LRFD-3.0.xls
    - HS_Str-AISC-Metric.xls
    - HS_Str-AISC-SHAPES-3.1.xls
    - HS_Str-AUST.xls
    - HS_Str-AUST-05.xls
    - HS_Str-BS.xls
    - HS_Str-Chile-2000.xls
    - HS_Str-China-2006.xls
    - HS_Str-CISC-7.2.xls
    - HS_Str-CISC-8.1.xls
    - HS_Str-Euro.xls
    - HS_Str-EURO-OTUA-2002.xls
    - HS_Str-Japan.xls
    - HS_Str-Japan-2005.xls
    - HS_Str-Russia.xls
    - HS_Str-SAISC-2002.xls
  - When bulkloading any of the steel libraries, it is necessary to first bulkload the corresponding Structural StructCrossSections workbook, also located in the AdditionalDataFiles folder.
  - A Cardinal Point control similar to the one in the Structure task is available for placing Rich Steel parts.
  - The Rich Steel parts allow modification of the Begin Cap, End Cap, Begin Face and End Face Port Cardinal Point values for interactive placement. The Flex Port can be modified and can be connected to the geometry of other parts, but it does not actually serve as a connection point to other Ports. Only the Neutral Port at the center of the length of steel cannot be moved or rotated. Most of the other Port manipulations, such as Offset and Rotation, are for use in assemblies.

        - Selecting the part and then pressing Ctrl + Shift + P will display the locations of the Ports.

    - With the Rich Steel parts, the Begin and End Cutback angles can be set during or after placement. With the Rich Steel, it is possible to have compound Cutback Angles by setting different Begin and End Cutback Anchor Points. The Rich Steel L Section provides the option of Cutback or Snipped Steel End Treatment. When selecting Snipped, the Snip Anchor Points are read-only.

        - The Cutback and Snipped parts can be used as supporting steel for placing Supports by Structure.

- A 3rd Party Application is now available that will provide an interface to the Lisega LICAD Program and other similar ones. It will allow the designer to design a Support, then automatically instance all the parts into the SP3D model.

    - **IMPORTANT**: To use this command, your administrator must set the required security setting in order for the .NET assemblies to work on a network share. If you do not do this, you will receive the following error message: "Not able to get the ThirdParty Plugin from the Catalog Rule or Default Rule." Refer to [*Smart_3D_Programming_Resources_Installation_Directory*]\CommonApp\SOM\Doc\3DDOTNETGuide.pdf for more information. You must install the Programming Resources to access this document.

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Hole Management

- Hole traces can be placed under any folder in the System hierarchy.
- Hole cuts are automatically placed as children of the penetrated structure. They cannot be manually relocated.
- Model Data Reuse, Model Data Transform, and Delete Optional now support Hole Management.

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HVAC

- Spec-0 and Spec-1 differ in relation to Duct Parts. Spec-0, as routed, is one part unless a component is inserted or existing features are modified. Spec-1 creates a part for each feature.
- While routing from the cell division, select the HVAC Nozzles in the locate filter combo box then select the required cell division.
- For Duct Spooling, Duct break points should be inserted after selecting Duct connections.
- To insert a Split on a Turn Feature, the user should select Single Split as the Split Mode.

Interference Checking

- The delivered Clearance Rules are for example only. Users should create/modify the Clearance Rules to match their own construction and design standards. This can be accomplished by taking advantage of the example clearance rules found in the IFCRule.xls workbook. The delivered example rules reflect the combination of objects that participate in the Smart 3D Interference Checking. If the user desires to include an aspect other than the ones already provided in the rule, then this can be done by replacing the aspect column field with the desired aspect to be checked. Any changes that you make to the clearance rule will not take effect until the new reference data is bulkloaded to the Catalog Database and IFC has been restarted or has had a Warm Reboot.

- If additional processing rules that exclude certain combinations are required or if rules that automatically set properties on IFC objects during creation are required, these can be achieved by modifying the InterferenceRules.sln file. This rule is provided with the example rules in the Smart 3D Programming Resources installation.

- If the Smart 3D Plant name is too long to display in the Plant drop down box on the "Interference Checking Settings" dialog box, hover over the drop down and the complete plant name will be displayed in a tool tip.

- IFC works on an intersection basis and not a volume occupancy basis. If an object is located completely inside another, no interferences will be generated. For example, Object A is a box measuring 2 meters x 2 meters x 2 meters. Object B is a box measuring 1 meter x 1 meter x 1 meter. If Object B is located 100% inside of Object A, no intersection will occur between the box faces, and consequently, no interference will be generated. Note: Clearance interferences will be considered if one is configured to Check Object A against Object B.

- HVAC has the ability to create a very complex Duct Part that could potentially bend back onto itself and create an intersection. Upon the completion of the part, the series of turns and straight features result in one part. Therefore, because a part cannot generate an interference with itself, Smart 3D does not return an interference object for this intersection. This is similar to the behavior of Shapes in Designed Equipment. When two Shapes that are children of the same Designed Equipment intersect, IFC objects are not created.

- The IFC completion time estimator is strictly an estimate and should not be heavily relied upon because of the complex nature of the relationship database created by Smart 3D. The value presented is intended to be helpful but may not always be accurate due to changing machine/network resources, as well as ongoing fluctuations in activity on the IFC Server.

- You need to be administrator of the system to configure and invoke the Smart 3D Interference Service.

- Users should select the Site Server and Site Database on Configure Interference Service dialog first and then start Smart 3D Interference Service.

- To change the Site Server and Site Database (and other selections) on the Configure Interference Service dialog, one needs to stop the Smart 3D interference Service.

- When uninstalling and reinstalling Smart 3D software, the user needs to reset the Log On Account Password for the user who needs to start the Smart 3D Interference Service.
- Users can check the Event Viewer log for explanations for Smart 3D Interference Service failures.

- If the IFC server was rebooted without stopping the IFCNTService, the Interference Detection process needs to be invoked manually.

- When Remote IFC has started to check for clashes with Point Cloud objects, the Smart 3D Interference Detection Service should be started from the command window using the following string: “IFCNtSvc.exe /Auto /0”.

- Warm Reboot option is the default option provided in Project Management when the user changes any IFC setting and starts it. This enables the user to restart the IFC process over again without losing all the approved fouls or notes on the markers. However, if the user wants to restart the IFC process completely then that user should use the custom command, “GSCADFoulCheckCmds.cSetIFCForRestart”. Note that when using the custom command all the notes on the markers and approved fouls would be deleted from the database.

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Materials Handling

- It is recommended to create the coordinate system for a conveyor, such that the length of the conveyor belt is always in the positive X (East axis) direction.

- It is mandatory to name the module with the same name as the parent system while copying it to the catalog for successful placement using the Place Belt Components command.

- If users intend to rename the parent system created through Place Belt Components command, then they need to rename the parent system in 3D first. Then they can rename the respective system in 2D via changing the Layer Properties.

- Layout of Equipment, Modules, and Trestles on an Overland conveyor can be done only with Place Belt Components in 3D command.

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Model Data Reuse / Model Data Transform

- Before running Model Data Reuse, it is recommended that you run Synchronize Model with Catalog. You should also run Database Integrity and Database Clean to ensure integrity, and clean out the To Do List as much as possible. These actions combined will increase the rate of successful objects transformed.

- With the Model Data Reuse command, you can copy large amounts of data and use the data again. For example, this command is useful when new plant systems or sections can be based on existing models. The Model Data Reuse Wizard can be found under the Tools menu in Project Management.

- Users can copy systems between models even if they are different site databases. A shared catalog is recommended. However, the compare catalog functionality now compares catalog schemas and pipe specifications prior to the copy operation. This helps the user decide whether or not they want to proceed with the copy operation based on the results of the comparison.
- There are two different methods to aid users with selection of source data. The first method allows the user to manually select source systems to copy, and the second method allows the user to select a source filter that will select source objects based on the filter definition.
  - Source System selection method - Users can select a single source system or multiple source systems to be copied. Use Ctrl+Click to select multiple source systems. If a user selects a single source system or multiple source systems under the same parent, then only a single destination parent system can be chosen.
  - Source Object by Filter selection method – Users can select filters defined on the System, Volume, Permission Group, Object Type or Properties tab. When a filter defined on the Volume, Permission Group, Object Type, Properties, or a combination of two or more of these tabs is selected, the software searches upward in the Workspace Explorer hierarchy until it locates the system that is the lowest common parent for all the copied objects. This search is conducted at processing time or when the MDR Wizard is “preparing to copy”. Since the lowest common parent is not known until processing time, a logical place holder for this system exists on the Destination Page of the Model Data Reuse Wizard. Please see the Model Data Reuse section in Project Management Help for further information.
- There are five Transformation modes for the Model Data Reuse command.
  - The “None” transformation mode is the same as the “Paste in place” option when using the common Copy/Paste command.
  - The “Move by relative distance” mode is the same as copying a set of data and moving it to specific coordinates in the model. Please note that the data will always be moved with respect to the Global coordinate system. If the area you are moving was modeled with respect to a rotated coordinate system, the results of the move may not be as expected because the software calculates the move based on the orientation of the Global coordinate system, which is 0 degrees.
    - The “Rotate” mode allows the user to copy and rotate data.
    - The “Mirror” mode allows the user to copy data around a plane. It is the same as using the common Mirror Copy command with the “Point to Mirror About” destination mode.
    - The “Transform by coordinate system” mode allows the user to copy, move and rotate data from one coordinate system to another in one step.
  - For three of the five Model Data Reuse transformation modes, a reference coordinate system must be placed in the Smart 3D model.
    - For the “Rotate” mode, the Z-axis is used as the axis of rotation. Therefore, it is recommended that the Z-axis of the “Axis coordinate system” be vertical. With the Z-axis vertical, the orientation of the X-axis does not matter. If you are copying data between models, the “Axis coordinate system” must be selected from the source model.
    - For the “Mirror” mode, the ZX plane or the ZY plane is used as the plane of reflection. It is recommended that the orientation of the Z-axis is vertical for the

"Plane coordinate system". If you are copying data between models, the "Plane coordinate system" must be selected from the source model.

- For the "Transform by coordinate system" mode, the orientation and the origin of the "From coordinate system" and the "To coordinate system" are used. Therefore, the orientation of these two coordinate systems does matter. For example, if the "From coordinate system" is placed with a bearing of 0 degrees and the "To coordinate system" is placed with a bearing of 45 degrees, the source objects will be moved to the location of the origin of the "To coordinate system" and rotated by 45 degrees. Please note that if you are copying data between models, the "From coordinate system" is selected from the source model, and the "To coordinate system" is selected from the destination model.

- A Review MDR Results command is available on the Tools>Utilities menu in Smart 3D. This command allows the user to graphically review the results of a Model Data Reuse operation. First, the user should browse to the location of the MDR log file created in %temp%. Then, all objects copied, successfully or not, are listed in the dialog. Once an object is selected in the table on the dialog, it can be highlighted in the Smart 3D model by pressing the "Add to Select Set" button. Using this command, the user can:
  - Easily find the source object copied and the resulting destination object
  - Filter objects that failed to copy
  - Filter objects by discipline
  - Create a report using Microsoft® Excel that contains additional information about the items copied during an MDR operation, such as system hierarchy path and OID.

- To troubleshoot the results of a Model Data Reuse operation, the user can perform the following steps:
  1. Check Database Integrity reports.
  2. Check the To Do List in both the source and the target models. For example, missing catalog data or missing symbols would be reported in the To Do List.
  3. Attempt to copy/paste/mirror/rotate the source object(s) using the common commands found in Smart 3D. Check the To Do List for any new entries.
  4. Use the Review MDR Results custom command to find any of the original objects or target objects quickly. All objects copied, successfully or not, are listed in this dialog. It also produces a report that will list the full system hierarchy path to the objects in question.

- After running Model Data Reuse or common Copy/Paste commands in a model, source objects may differ from the destination or post-transformation objects in some cases. Model Data Reuse and common Copy/Paste commands do not simply copy source objects to a new location or orientation. These commands actually analyze the source objects and recreate them in the new location using the catalog and rules that are in place at the time of the copy. Users often modify their catalog and rule sets over time. If there have been changes since the time the source objects were modeled, there may be differences seen in the destination or post-transformation objects caused by the recalculations performed while processing Model Data Reuse or common Copy/Paste transactions. In addition, if the source objects were modeled in an older version of SmartPlant 3D, differences can be attributed to improvements in Smart 3D solvers or changes to application commands. In some cases, source objects are not updated during version upgrade by design. If it is necessary to recompute or update source object(s), the Update Selected Objects custom command is available for this purpose. The instructions for running this command are given below:

    1. Select the source objects that should be updated graphically or through Workspace Explorer. We recommend selecting no more than 10 objects at a time due to memory limitations.
    2. Select Custom Commands from the Tools menu.
    3. Press the Add button and enter SP3DDebugCommands.CUpdateObjects as the Command ProgID.
    4. Enter Update Selected Objects for the command name and press OK.
    5. Select the command from the list and press Run. There will be no progress bar for this process. It may take a few minutes to run depending on the select set used.

- Do not modify or reorganize source or target systems while MDR is running.

- It is a best practice to close Project Management environment after each MDR Operation.

- Model Data Reuse can be used to copy data within a model, across two models that exist in the same site, or across two models that exist in different sites. With the unification of the S3D product, it is now possible to copy data across models of different types (i.e. from a MHE model to a ship model).

- Model Data Reuse operations across sites that are harbored in different servers are allowed, but the servers may not belong to different database platforms. For example, data may not be copied from an Oracle site to a SQL site.

- If data is copied across models that point to different catalogs, additional instructions will appear in the Model Data Reuse Wizard to allow users to compare, view, and map the content of the source and destination catalogs. For additional information on these cases, please refer to the S3D Project Management Guide.

- Data copied with the Model Data Reuse command should find all necessary metadata in the destination required for the data's interpretation. Attribute data that does not meet this requirement will be cleared out to protect the intellectual property of the source model's owners.

- The Check Write Access command within the Model Data Transform command allows you to identify permissions or status issues and disconnections prior to executing the transformation. This information can be copied to the clipboard and it is recommended that it be carefully reviewed.

- Electrical cables have a special behavior with the Model Data Transform. The transformation of a cable is not implemented as a rigid transform, but instead is implemented as an update of the cable geometry to ensure that it remains connected to both the originating and the terminating devices.

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Molded Forms

- For the Molded Forms opening command, the Unified Sketcher has been implemented. The Opening Sketcher ribbon bar and behavior will be similar to the Structure opening command for a slab.

- Plate systems support top-down design. Start with the whole large plates and add information until smaller leaf parts are fully defined. This also supports an iterative design approach.

- Plate systems are defined by the “molded surface” of the plate.

- Molded conventions are properties that determine which side of the line/plane represents an object’s (e.g. bulkhead plate) thickness. These properties are assigned defaults in the reference data and can be modified later in the design process.

- Design seams define locations where structure material, thickness and other system properties change within a structural system.

- Stiffener Profile Systems are structural shapes that can be connected to plate faces or edges to stiffen the plate. They are commonly referred to as “Profile Systems”. Members are structural shapes connected only at their ends to other structure. They can be standard (rolled) members or design (built-up) members.

- Stiffener systems are represented by a landing curve and cross section. The landing curve represents the “molded line” location of the stiffener on the parent plate system (within the plate system boundaries). The cross section (Location/orientation) is defined relative to landing curve (cross section oriented in plane normal to landing curve).

- Molded Forms opening uses the Sketch2D ribbon bar, similar to the Structure Task.

- Molded Forms plate systems, in Split Root System functionality, can use design and planning seams for splitting.

- Molded Forms systems support the copy/paste from-to functionality.

- Molded Forms plate function has enhanced the boundary list controls to support optional inputs and user input of offsets for boundaries.

- Molded Forms Objects are now supported by the Work Breakdown Structure (WBS) functionality. This allows Molded Forms Objects to be grouped in a Work Breakdown Structure. This WBS can be used to collect function, area, cost, or schedule information for a group of objects.

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PDS Reference

- The ability to reference PDS projects through PDS reference is no longer supported in version 2016. Users can use Reference 3D to refer the PDS projects which has enhanced capabilities like 'include/exclude' selected objects, load hierarchy etc. Users should ensure that all the PDS projects are de-referenced from Project Management before taking a backup from 2014/2014 R1. If a backup is taken with a PDS project referenced, the users cannot upgrade these databases to 2016.

- Re-reference the PDS projects using Reference 3D (R3D) as it is the preferred method to reference PDS models.

- All Smart 3D Drawings, Filters, surface Style rules, related modelled objects, etc. that had used or had relationships to the referenced PDS project will the out of date. The Drawings will need to be updated, filters and surface style rules modified, and object relationships re-created after re-referencing the PDS project using Reference 3D (R3D).

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Piping

- To change the option for Temperature and Pressure so that a Warning message is displayed instead of an Error message, the user needs to change an option in the Ten_Specs_SpecificationData.xls workbook in *[Smart_3D_Installation_Directory]\CatalogData\Datafiles* (or the equivalent spreadsheet for your catalog). The option is located in the Default Project Options worksheet under the UndefinedServiceLimitsRuleOpt column. User will need to change the option from 5 to 10, put an M in front of the row and bulkload the spreadsheet using the Add, Modify, Delete option. After this change, if the user routes without a Temperature/Pressure combination, a warning message appears that will have the option "Do not show this again".

- To setup default properties for Piping Systems, such as Temperature and Pressure, first create Piping Systems in the Systems and Specifications environment. After creating the Piping System, the user is able to set any default properties desired from the Default Property Values tab in the Piping System properties page. Any Pipeline created under a Piping System that has default properties defined will also have these properties defined.

- Welds are not named with a sequence number until the user runs the Sequence objects command.

- For successful insertion of a component with a maximum temperature defined, the pipe run must have a maximum temperature defined.

- Mating parts cannot be tapped.

- By default, Copy/Paste of objects will result in objects pasted in place. However, if the user wishes to move objects dynamically while pasting, that user can uncheck the “Paste in place” check box on the Paste Special dialog. When the “Paste in place” option is unchecked, Paste allows the user to dynamically select the placement point.

- The Flip command button toggles the ports available for the component that is being inserted. As each port is toggled, the component is oriented so that the selected port is aligned along the axis of the leg where it is being inserted.
- Piping Specialty and Instrument components that are defined in Reference Data can be inserted on a run by selecting <Specify Tag Number> from the type menu and entering a valid Tag Number.

- To apply an Offset between a Piping and a Structural Surface, select Tools > Options and select the SmartSketch tab to turn on the "Edges on solids" option. Note that a plane must be set for this icon to appear. If the plane is set to ‘No Plane’, the user will never see the “Edges on solids” icon for offset.

- To route sloped piping correctly, toggle the slope on and off while routing. If you need to branch out of a sloped pipe feature, start the route command and branch out of the pipe while routing. Do not place the component manually using the Insert Component command.

- When pasting piping features a parent run is automatically created for the feature. If you want the feature under a specific run, you will have to change the parent run after pasting.

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Piping Isometric Drawings

- The Drawings by Query method to produce drawings does not support the use of My Filters as the “What” or the “Where” filter.

- Several Drawing Filters are delivered in the default SP3D catalog to assist in creating Drawings by Query for isometric piping drawings. These filters are available in Catalog Filters > Default Filters > SP3D Drawing Filters > Types of Drawings > Isometric Piping. They can be used as the What Filter in the Setup for a Piping Isometric Drawings by Query. In addition, five packages are included in the default symbols share: Iso Pipeline, Iso Spool, Iso System, Iso Stress and Iso PenSpool. These packages point to the delivered Iso Styles and Drawing Filters. You can reference them in the setup for a Drawings by Query Manager component and provide a Where Filer. Refer to the Drawings and Reports Users Guide for more information on using Piping Isometric Drawings by Query.

- We recommend that users use a shorter path for the TEMP directory than the default location (ex. C:\Temp). Keying in %temp% in the address bar in Windows Explorer will show current location. This will help avoid problems which can occur when using very long filenames (109 total characters for filename and location).

- When a single user logs into both the Host and Satellite or more than one site in a Global Workshare Configuration, the Run Query operation will fail for all sites but the one the user logged into first. For example, if User A logs onto Satellite B, then User A will not be able to Run Query at any other location but Satellite B. The Run Query operation will fail at the Host or any other Satellite location. An hourglass will appear for a couple of seconds and then disappear. No data will be created under the Drawings by Query Manager component. (Note: This problem exists even after creating new Drawings by Query Manager components.) This occurs because the Run Query operation internally creates a user filter in the My Filters folder causing ownership of the My Filters folder to transfer to the user updating the drawings. Therefore, due to the lack of permission on the My Filters folder at any alternate locations, the Run Query will fail. To work around this problem, move or transfer ownership of the "My Filters" folder to the appropriate site and create new Drawings by Query Manager components.

- Changes to Piping Isometrics were implemented that defines criteria for a valve operator to appear on an Isometric Drawing and for the spindle direction be called out on Isometric Drawing. Valves defined in catalog data as having an operator and whose symbol is implemented with an operator occurrence will have the spindle direction called out on the Isometric Drawing. This also applies to turn features, such as Pressure Release Valves and Angle Valves. For On-the-fly type Instrument Valves, the Valve Operator Type must be defined on the Material Control Data tab of the Part property page for the spindle to appear on the Isometric Drawing. Also, for On-the-fly type Instrument Valves, the Commodity Class needs to be set to Valves on the Definition tab of the Part property. If this is blank, the spindle will not appear on the Isometric.

- The Piping Isometric WBS Item Continuation Label now ignores the WBS Purpose setting when placing a continuation label. However, this only applies to Iso Drawing, Iso Drawing – Tracing, Iso Drawing – Stress, Iso Drawing – Fabrication, and Iso Drawing – System Purpose Types.

- Users may now set the decimal separator to a character other than a period in their regional locale settings for use with the Isometric Options Browser. Options that require floating point values are always stored in the Style XML with a period decimal separator, which is a restriction of the Isometric Style format. However, these values are now displayed and edited in the Isometric Options Browser per International software standards. There are some caveats to this behavior, however.

    - For example, individual options, such as WeldList.UserDefined.VerticalSpacing, are held in a "text" cell. Entering an invalid value for one of these options will result in a message explaining that the value cannot be converted, which is displayed in the status field of the browser. Then, the cell is reset to its original value.

    - For numeric attributes entered in any of the option tables, such as DrawingFrame.Attributes.XPos or Drawing.Definitions.XPos, minor typographical errors in data entry can result in a value that was not intended. For example, presume the locale's decimal separator is a comma. If you change the value of Drawing.Definitions.XPos from "76,00" by entering the value "5.00" (note the period), the result will be "500,00" with no warning to the user.

- Changes have been made when a user updates an Isometric that specifies a Coordinate System on the Drawing Properties Style tab.

    - If a Coordinate System specified by the user in the Style tab is not found, the Isometric will update in error state and the log file will relay this information to the user. Previously, the Global Coordinate System would have been used when updating the Isometric.

- If there are multiple Coordinate Systems with the same name as the Coordinate System specified by the user, the Isometric will update in error state and the log file will relay this information to the user.

- All SmartPlant 3D specific options now appear under a separate node (S3D) in the Options Editor tree.

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Planning

- Planning functionality is supported by Model Data Reuse/Model Data Transform.

- The Sequence tab has been removed from the Assembly properties page. Use the Sequence command on the vertical toolbar, then select the assembly.

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Project Management

- Ensure that enough disk space exists on the database server before creating the databases.

- Backup the various databases properly before deleting to ensure recovery.

- Stop the IFC Database Detect process before installing any new Smart 3D software.
- Ensure that users have the required permissions on the locations to which the various files/databases will be stored during the backup/restore process. Note that users (as well as accounts used as log-in for Oracle services) need to have write permissions on the folder under which the .bcf file is stored.

- The Modify Database and Schema Location form should be set with the server name that was used while creating the Databases. That is, if a fully qualified name is used while creating databases, specify the fully qualified name on this form. If the databases are created with a name which is not fully qualified, use the same on this form.

- If a plant is deleted from Project Management environment after backing it up, users cannot restore the plant directly with the Tools > “Restore one or more plant databases from backup” command in the same site. To restore the plant, first restore the site through Database Wizard (after deleting the databases using SQL Server Management Studio or tablespaces and users in Oracle). Then, restore the plant from the Project Management environment.

- There are three levels of access permissions that can be granted to a user/group in a Permission Group. They are "Full Control", "Write", and "Read".

- When specifying the symbols share path, do not place a slash at the end of the path. This can cause problems for those environments that write information to the symbols share.

- During creation of Oracle schemas from delivered templates or restored from backups, some systems might report errors such as: “ORA-31684: Object type … already exists or ORA-39082: Object type … created with compilation warnings.” These errors can be ignored.

- During the restore of an Oracle Database some systems might report errors such as “ORA-39060: table(s) dropped because of conflict with master table”, this error is caused due to unexpected termination of the Restore process.

- Before setting up a Global Workshare Configuration (GWC), consider all possible future scenarios, such as how individual locations might be added or removed, as well as where most of the design work will occur, and define the best strategy based on this analysis:
    - For Oracle Global Workshare, it is recommended to configure the location where the majority of work is done at the Hub. Keep in mind that the Hub cannot be removed individually from a GWC.
    - For MS SQL Server, it is recommended to use dedicated distributors at locations where a significant amount of work is done.
    - Multiple Plants (in the same site and site schema) can be configured for Global Workshare provided they use the same set of locations as the first GWC.
    - When configuring Global Workshare, ensure that all processes, such as IFC or Drawing Batch server, are stopped and inactive before running the “Duplicate Plant for Workshare” command and no one is connected to the plant until the setup is completed.
    - After running the “Consolidate Plant from Workshare” command on Oracle Global Workshare, copy the generated scripts to the GoldenGate directory on the corresponding database servers and run the consolidation batch scripts.

- In Smart 3D, certain inconsistencies will arise during the updates that occur in GWC. The Duplication Synchronization Service (DuSS) addresses these issues to keep the model data accurate at all locations. DuSS need to be configured at each geographical location and added on Database tab of Plant Properties form. The DuSS service is not running by default. Follow the below steps to install and configure DuSS:
    1. Log on as an administrator on the local machine.
    2. Add [*Smart_3D_Installation_Directory*]*\Core\Runtime* to the PATH environment variable
    3. Open a command window as administrator
    4. Use the cd command to switch to [*Smart_3D_Installation_Directory*]*\ProjectMgmt\Middle\Bin*
    5. Execute the following statement: “DuplicationSyncService.exe –I”. (Note: Later on, you can execute “DuplicationSyncService.exe –u”” to uninstall it).
    6. Close the Command window
    7. Pull up properties for the DuSS Service and go to the Log On tab.
    8. Provide a user who has local machine administrator and administrative access to Smart 3D databases.
    9. On the General tab set the Service to Automatic and then Start it.

- When the Database Integrity command is interactively executed, a text file is generated named DbIntegrityCommandLine-PlantName.txt. The content of this file is a command line task that can be scheduled using the “at” command or using Windows Scheduled Tasks. Options used in this command line are identical to the options used when Database Integrity was interactively executed.

- For cleanup of Oracle schemas and users, when deleting Smart 3D Sites and Catalogs, Oracle DB Console or SQL scripts can be used. It is also possible to remove them using the SP3DOracleDBToolsWizard.exe located in *[Smart_3D_Installation_Directory]\ProjectMgmt\Tools\Bin*.

- The SQL Server Collation Utility under Start > All Programs > Intergraph Smart 3D > Database Tools provides a way for customers to specify the default collation that should be used by the Database Wizard, Project Management, and the Bulkload utility for creating databases. This utility creates a copy of the delivered schema templates (AppRepos.dat and CatalogDB.dat) with the desired SQL Server collation. Any site, catalog, model, and reports databases and schemas created from these templates will use the specified SQL Server collation. Note: Only homogeneous configurations are fully supported, not heterogeneous.

- The Update Custom Symbol Configuration command provides a way to create and update the custom symbol mapping file, as well as the mapping file for delivered symbols. The custom symbol mapping file contains the program ID (ProgID) and part class ID (CLSID) for each of your company's unique symbols. When a custom symbol is added or updated, you must run the **Update Custom Symbol Configuration** again to update the custom symbol mapping file. This command decreases the need to register custom symbols on the local machine. When a custom symbol is referenced in the model, the software will look in the CustomSymbolConfig.xml file to see if an entry is present for the symbol.
    - This command can be run by right-clicking on the catalog node in Project Management, which will create the CustomSymbolConfig.xml (used for Custom Symbols) and the SystemSymbolConfig.xml (used for delivered symbols in the \\<server_name>\...\SharedContent\Bin folder). These two files are located in the \\<server_name>\...\SharedContent\XML folder on the symbol share.
        - This method cannot be supported for certain custom symbols after the command is completed, review the log file if you have any symbols in the “Custom Symbols” folder that are not supported.
    - It is highly recommended that users place their customized symbols in the \\<*server_name*>\...*SharedContent**Custom Symbols* folder. The \\<*server_name*>\...*SharedContent**Bin* folder should be used for Intergraph delivered symbols only.
    - Unused symbol files should be renamed in such a way that the original extension is not retained. For example, BallValve.dll would need to be renamed to BallValve.dll.**old**, instead of BallValve_old.dll.

- Reference 3D (R3D) functionality facilitates the attachment of external 3D data as a reference model. Such external data might have been published from another Smart 3D model or from certain third-party models. The data from external models is expected as a set of graphic (.zvf) and data (.xml or .drv) files. Using R3D, you can attach and orient (position, rotate, and scale) the reference model relative to the active model so that you can view the reference model data graphically and model against it. In addition, users can inspect the R3D model objects' properties and view their names through ToolTips. You can extract simple General Arrangement drawings showing relative positioning of objects from R3D models, with limited annotation. This command significantly augments the current Global Workshare solution, as it does not require replicated databases.
    - When attaching an R3D model to the plant configuration for the first time, the R3D schema is loaded and the object hierarchy (BOC) is updated. This can take around 20-30 minutes to complete. If the user wants to attach a new R3D model to replace the currently attached one, the new R3D Model should be attached before detaching the old one. This will save the user time because the schema does not have to be removed and re-created.
    - Reference 3D Models are indicated as Out-Of-Date if the timestamp on the graphical file(s) are changed. If new files are added to the Reference Model folder, the user should update the R3D Model manually.
        - To force reimport of R3D Schema, the user should modify the mapping file version.
        - To improve performance while attaching R3D Models, it is recommended to comment (using "!") properties that are not required in the mapping file.

- For 2016, Project Management hierarchy icons have been updated. Refer to the *Project Management User's Guide* for more information on what these new symbols represent.

- To Do List (TDL) item descriptions have been improved to be more meaningful messages. However, these descriptions will not update automatically in databases with TDL items that are being upgraded to 2016. Once the TDL item is manually updated, or once the object associated to the TDL item is modified, then the TDL item description will reflect the newer, more meaningful message.

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Reports

- Smart 3D supports Microsoft Excel 2013 (32-bit). Excel has a file format and extension (.xlsx). However, the delivered Smart 3D report templates still use the old format and file extension (.xls). Excel implicitly runs in compatibility mode with .xls files. For report templates created in Excel with the .xlsx extension (created outside of the Smart 3D software), only machines with Excel 2007 or later (32-bit) will be able to process (open, edit, or update) the report.

- When saving report output on a Windows 7 machine, a sharing violation may occur and the user may be asked to save a temporary file. This is due to third-party virus protection software. To work around this issue, load the latest service pack or hot fix for Office 2013 and check the online support for your virus software to see if there is any additional information.

- To ensure that Microsoft Excel spreadsheets display properly for Office 2013 (32-bit) users, click the Microsoft Office button to access Excel Options. Go to the Trust Center category and select the Trust Center Settings. Select the Macro Settings category and check “Trust access to the VBA project object model”.

- The Reports handle auto-register of user added macros. If a user adds the macro.xla files at delivered macro files location, these macros are automatically copied to the Microsoft Office Add-ins directory at runtime and registers them as Add-ins in the Microsoft Excel such that these custom macros can be used by the Reports.

- Smart 3D reporting is implemented to report all objects that are queried by the Catalog Filters in the Smart 3D Report Filters folder. The delivered filters currently query all objects in the database, regardless of their status or reporting requirement.

- To exclude certain objects from reports based on their properties (for example, objects with a Reporting Requirement = ‘Not to be Reported’), it is necessary to modify the delivered filters. For the applicable report, modify the corresponding filter in the Catalog Filters > Default Filters > SP3D Report Filters > Types of Report > … folder. Using the delivered filter definition, add another criterion on the Properties tab so that only Object Types with the “Direct Property of Object Type” set in the Standard Category and the Reporting Requirement = ‘To Be Reported’ will be returned by the filter.

- Reports are saved in the Catalog using the “Copy to Catalog” option in the Drawings and Reports environment.

- Labels are saved to the Catalog from the Label Editor in the Catalog environment.

- When a change is made to the Catalog Database schema or Site Database schema, the Reports database must be regenerated using the Regenerate Reports command in the Project Management environment.

- When creating a custom report in the Drawings and Reports environment, select File > Save Report Template before exiting Edit Template mode in order to retain the changes to the report.

- Reports that use a filter should point to a filter that is created as either a Catalog filter or a Plant filter. If the report is used in multiple plants that access a shared catalog, the report should point to a Catalog filter. If the report is used only in one plant, the report may point to a filter that is either a Plant filter or a Catalog filter.

- Modified File Locations are saved in the session file. If a user opens a session file that was saved by another user and the Report Output points to a directory where the user does not have access, reports will not be created. File Locations are modified in Tools > Options > File Locations.

- Edit Tooltip on the Tooltip tab saves modified tooltips as My_XXXX_Label. Permanent modifications to delivered tooltips are made using the Edit Label button found on the Define Label dialog in the Catalog environment.

- Reports require that the following packages in the Site and Catalog databases be synchronized:
  - CORE
  - COMMONSCHEMA

- COMMONAPP
- PRJMGT
- UDP

If these packages are not synchronized, Report Database generation will fail and the following message will appear:

> "Cannot correctly generate the Reports Database because of a metadata package version mismatch between the Site Schema and the Catalog Schema. Site Schema has <packagename> version <packageversion>, where Catalog Schema has <packagename> version <packageversion>. See your Administrator to resolve this migration issue."

- Reports may be used to find an object in the Graphic view. A Custom Command is delivered named "**SP3DFindObjectByReport.FindObjects**". This command allows you to select a previously run report, highlight a row in the spreadsheet and the selected object will highlight in the graphic view. Note: This command only works for reports that return OIDs (Object IDs).

- When new components are added using the Bulkload utility, their generated identifying interfaces must be added in the appropriate files in the report query template, such as Material Take-Off - Component.rqe file, in order for the new components to be listed in the report. If non-English characters are used in the component names, the rqe file must be saved in UTF-8 format

- If the user tries to print reports created in the Drawings and Reports task on a workstation that does not have a printer set up, the command seems to run, but no document will be printed. No warnings are generated for the user.

- An embedded report in a drawing does not show borders for text cells if the border is not explicitly added in the Report Template. You can edit the Report Template to add or delete the borders of the cells to be able to see the desired borders in the embedded report.

- The behavior for Repeatable Labels inserts a new line character after the attribute data of each child object. If a line break is required, the user can open the Label Editor, place the cursor just before the </B> and press the Enter key. As a result, the </B> will move to next line and therefore, the attribute data will be displayed in the next line.

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SmartPlant Enterprise® Integration

- When upgrading a plant configuration, please note that the Smart 3D Upgrade Guide needs to be followed with regards to the Integration steps. Specifically, ensure the Symbols (SharedContent)\XML directory is not overwritten. As one of the last steps, ensure the Upgrade Schema command is run. This Upgrade Schema command will upgrade the necessary files in the Symbols (SharedContent)\XML directory to the latest release (i.e., 2016). As part of this process, the user will need to ensure the files in the SharedContent\XML directory are writeable before beginning that process.
  - Note: As part of the standard process, we always recommend making a backup of the SharedContent (Symbols) \XML directory before the process begins.

- In order to easily select multiple objects whose Design Basis have been deleted, the user can create a filter. To create this filter using your present filter, while in your model select Tools-> Select by Filter.

    1. Highlight "Plant filter" and select the New Filter button on the ribbon bar.
    2. The New Filter Properties dialog box will appear. Enter a name for the filter, and select the Property Tab. Under the property field, select the drop down and select "More". The Select Properties dialog box will appear.
    3. Enter the following Values:
        a. Object type used as the basis: Equipment under Equipment & Furnishings
        b. Relationship: Direct Property of Object Type
        c. Display properties in this category: Standard
    4. Next highlight Correlation Status and select OK. The new Filter properties dialog box will appear, then select "=" under operator, and Correlate Design Basis Deleted under value, then OK. Select the filter you created once the Select Filter dialog box appears, and then select OK. This will highlight all objects in the model whose design basis has a Correlation status of "Design Basis Deleted." Once highlighted, select the delete on the ribbon bar, or you can choose the individual items to delete.

- To better maximize memory during S3D model updates, processes named JEFHelperExe, JEFExpandDataExe and JEFDisplayFileExe run as separate processes during the update. If the user clicks in the window while the separate process is running, they will see an Intergraph Smart 3D dialog appear which states “An action cannot be completed because a component (SmartPlant 3D JEFHelperExe) is not responding. Choose “Switch To” to active the component and correct the problem.

- For publishing to SmartPlant Foundation (SPF) of any other document type than Smart 3D Model Drawings, the user must populate the following data on the WBS tab: Document Type, Document Style, Discipline and Allow Publish. These same settings are required to be populated before the Revise option is available to the user as well.

- For updating of Smart 3D Model Drawings in an integrated environment, the domain user logged into the client must exist in SPF as a valid user.

- The number of objects published from Smart 3D can become so large that the normal update and publish processing uses up the resources such as memory and address space on the client and server computers. To address these resource issues, Smart 3D now supports the concept of using a cache to keep track of objects that have changed and to only publish these objects. Thus, the software avoids processing objects that have not changed. The mechanisms for doing this are called Parallel Update, Delta Publish, and Auto-Scoping. These processes are not visible to the user. These require that when you publish a Smart 3D model, you must now enable the Scheduler and Loader in SmartPlant Foundation to make the Smart 3D model data document retrievable. The load, consolidate, and merge tasks must complete successfully before the Smart 3D model document can be retrieved. Both Parallel Update and Delta Publish can publish larger documents by automatically splitting the data into smaller sections in Smart 3D. If the published data is split into smaller sections, the document version number increments for each section. For example, if you publish a document that is split into four smaller sections, the Version field contains all resulting version numbers. The last version contains all of the updated information from the publish.

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Space Management

- To use the Merge Volumes or Create Volume Bound by Spaces commands, the user must have write permission to the sub-volumes selected.
- In sketch mode, multiple selections are possible by using the Ctrl key. This works the same as selecting multiple objects in the graphic view using the Select command.
- It is better to organize the parent folder and its sub folders in the same permission group. This helps the user to restrict the permissions for the group of folders and volumes.
- Currently, the user is not able to move or rotate Volumes by two points and Volumes along a path when placed on a Grid System unless the Associative point is removed.
- To use the "Place Drawing Volume by Selection" command, the user must first go to the Drawings and Reports task to place a Drawing View. Drawing Views can also be placed from respective graphic tasks using the Drawing console option provided in the Tools menu.
- Once a Grid is selected using the Place Volumes by Grids command, all X, Y, and Z planes are automatically highlighted. However, individual planes can be deselected by pressing the CTRL button and selecting the planes from the Workspace Explorer or from the graphic view.

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Structural Analysis

- Create an Analysis Model before defining Load Combinations or Boundary Conditions.
- Multiple Analytical Models can be created.
- The Members System folder contained in the Analysis Model will not have anything in it now that multiple Analysis Models can be created for the same structure (for various conditions). To review the members included in a particular Analysis Model, select the Analysis Model from Workspace Explorer and right-click to the “Select Nested” option.
- Only Loads belonging to a Load Case that is part of a Load Combination definition are written to the CIS file for export.
    - A mapping file must be used if the third-party analysis program you are using calls the section standard by different name from what is delivered with the Smart 3D mapping file. For example, STAAD calls its section standard 'AISC EDI Standard Nomenclature'. Therefore, the line in the mapping file that maps these two Section Standards must be changed from <Standard name="AISC-LRFD-3.1" externalname="AISC-LRFD-3.1” externalorganization="AISC" externaldate="2002" externalversion="3.1" /> to <Standard name="AISC-LRFD-3.1" externalname="AISC EDI Standard Nomenclature" externalorganization="AISC" externaldate="2002" externalversion="3.1" /> in order to get the section sizes updated.

- When placing multiple loads on multiple members, use Position as Absolute. In this situation, Relative position cannot be used easily. Changing the view to Elevation or Plan while placing loads may also help.

- Members placed using an offset value that creates the actual part geometry away from the placement axis will result in the analytical member location not matching the physical model. The analytical model uses the member axis and ignores the offset of the actual part. Members should be placed with this in mind, therefore keeping offset values to a minimum.

- When creating a new Analysis Model, make sure to apply the correct filter in the Analysis Model Properties. If the Analysis Model is without a filter, no boundary conditions can be applied to the supports.

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Structural Detailing

- Assembly connections drive automatic generation of the following based on catalog rules:
  - Slots and Clips/Collars
  - Chamfers
  - Profile End Cuts
  - Welds (physical connections) Manufacturing parts can be updated based on changes in Molded Forms, Structure and Structural Detailing via Manufacturing Service Manager (MSM)

- Trim operation uses part (not system) geometry and is not the same as bound (Molded Forms operation). Trim removes overlapping material between bounded parts and their bounding parts which creates exact 3D solid part geometry. There must be contact between a plate part and its bounding objects for trim operation to succeed and surfaces must contact entire thickness of plate part.

- Delete Optional (Copy Paste, Mirror Copy and Model Data Reuse) & Model Data Transform are now supported for detailing objects.

- Improved the handling of physical connections for non-overlapping geometry, split functionality & To Do List elimination of “member of custom assembly deleted” for deleted Physical Connections.

- Welds can be created between plates, stiffeners, and members to slab objects.

- “Profile Straking Seam” & “Beam Placement” commands are removed from the Structural Detailing Task.

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Structural Manufacturing

- Manufacturing parts can be update based on changes in Molded Forms, Structure and Structural Detailing via Manufacturing Service Manager (MSM).
- Structural Manufacturing task integrates with validated 3rd-party programs via XML files.
- The Part Monitor provides a graphics view and text file check of selected object prior to executing Output Manufacturing Data for fabrication.
- Manufacturing Service Manager tracks status of manufacturing parts. If changes have been made in Molded Forms, Structure or Structural Detailing, the manufacturing part is out-of-date. The manufacturing parts can be manually updated using the Manufacturing Service Manager.
- Manufacturing allows the creation of Panel Assembly / Welding parameters. Users can define a panel consisting of different plates, with different thickness and orientations, to be grouped together and treated as a single plate for cutting.
- Manufacturing, as part of a delivered integration of the managed export and the panel processing, has implemented a license (Smart 3D v11.0 Panel Cutting and Welding) for managed export and panel output.
- Structural Manufacturing added Work Breakdown Structure Support for Objects to be able to group manufacturing objects within a Work Breakdown Structure so users can collect function, area, cost, or schedule information for a group of objects.

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Structure

- Sketch 2D has been updated for plant designers. The orientation when sketching 2D slabs and walls is now based on the Active Coordinate System as defined in the PinPoint command. This is very helpful when sketching on rotated Coordinate Systems. Be sure to set the Active CS in Pinpoint to match the building or area being worked. Also, the grid lines can be displayed and used in the sketcher now without adding them as reference objects. Select View>Rulers to display the rulers for the Active CS in sketch 2D.
- Partial openings are not exported via the CIS/2 standard format.
- A ‘Centerline’ aspect is available for the delivered Handrail and Stair symbols. The existing ‘Detailed Physical’ aspect has been removed for both Handrails and Stairs. The Detailed Physical aspect was the same as the Simple Physical and was redundant. Users will need to run ‘Synchronize Model with Catalog’ to get the new Centerline aspect and graphics.
- When placing an Opening in a single Wall that has a path with parallel sections, the Opening also gets placed in the opposite Wall. Currently, the “Through Next” option is not supported for Walls. In order to place an Opening in only one Wall, you need to use the Cutting Limit with User Defined option and set the Cutting Depth equal to the Wall thickness.

- It is recommended that users place elevation and grid planes in the Grids task before you place members and Slabs.
- The supported member is the member that you are placing. The supporting member is the existing member in the model to which you are connecting.
- Frame Connections do not display in the model. However, connections can be located at the ends of the members and will highlight (as circles) when you move the cursor over them.
- If you are unsure of which Frame Connection type to use, you can select the "By Rule" option to allow the software to select a Frame Connection Type automatically.
- For fitted Assembly Connections, the system ignores the flange inside clearance values defined in the catalog for cope increment values set to non-zero. When Increment is non-zero, the system rounds off the cut lengths. User defined flange clearances are applied when cope increment is set to zero.
- Customized structural member cross sections can be created in SmartPlant 2D Symbols environment and bulk loaded to the Catalog database for use in the Structure task. In some cases, the connection rules must be changed in order to handle the new section types.
- It is recommended that the user place the Assembly Connections after finalizing the Frame Connections.
- The Assembly Connection, plate part, or feature goes to the To Do List when error conditions are encountered during placement or modification. The Assembly Connection is posted as "In Error" or "Out of Date" in the To Do List, under the following conditions:
    - Fitted Connection
        - The members are collinear.
    - Splice Connection
        - Members must be collinear and of the same shapes, mirror state and angular rotation. Circular non-flanged shapes are not supported.
    - Gusset Connection and Corner Gusset Connection
        - Members must be in the same plane and must be oriented properly. An error will occur if the brace is deeper than the supporting members. Reduce the brace size to fix the error.
- Error messages on the To Do List for Frame Connections/Assembly Connections are not always informative. Please use the above information to resolve any problems for the specific connection type and configuration, or review the To Do List Messages section of the TroubleshootingGuide.pdf file installed with the product at $:Program Files\Common Files\Intergraph\SmartPlant\Help.
- Assembly connections are automatically deleted when the supporting frame connection is changed to “unsupported” or when one of the members involved in the connection is deleted.
- During copy/paste of structural objects, make sure that appropriate related inputs are selected on the Paste Special dialog if required.

- The user can select grid elevation planes in the Workspace Explorer during the Define Plane step of the Place Slab command.

- The user can create multiple Sketch 2D Slabs on the same support plane or on multiple support planes by selecting one or more planes and sketching multiple closed curves in the Draft 2D environment during the Define Boundaries step of the Place Slab command.

- The user can only modify one slab boundary at a time. Modify a single slab boundary and Finish the command to update the geometry. Repeat the modification process for each boundary that needs changing.

- When the Slab has more than one layer, the Total Thickness field in properties is read-only. You have to switch to the Layer tab of the edited Slab to change each layer thickness. The Total Thickness will then be computed. If the Slab has one layer, then the Total Thickness is the Layer Thickness. In this case only, the Total Thickness field is enabled.

- The Opening should be well constrained to the Slab or Wall when placing it in the Draft 2D environment. Without any constraint, the Openings do not 'follow' the Slab or Wall.

- If you are placing an Opening that completely penetrates the item, the side (surface) selection is not important. However, if you are placing an Opening that does not completely penetrate the item, then the surface selection is important because the penetration depth is measured from the selected surface.

- Members can be more easily connected and positioned relative to other objects using the middle-mouse button to lock SmartSketch to the target member and then using the secondary relationship to position the member.

- Selecting the SmartSketch® middle-mouse button lock or the intersection between a member and a grid line will create a point-on constraint to the target supporting member in the grid plane. If the grid plane is moved, the supported point-on member will move along with it bounded by the ends of the supporting member.
- A command to place Fireproofing insulation on Structural Members is available. Fireproofing insulation can be placed by rule or by manual catalog selection. The catalog specifications have been established based on member type, such as Columns, Beams and Braces. They are also based on the number of hours the protection is needed. The user should go to Tools>Options and set the Units of Measure for Time to hours instead of seconds.

- Combined Footings do not place as expected on rotated coordinate systems or rotated columns. The user must modify the angle of the footing after placement to correct the orientation. The orientation must be checked for accuracy after MDR operations and corrected as needed.

- Members can be split using a “point” location, with relative and absolute distance options available. Group splitting is not available with the “By Point” option.

- Handrails can be converted to structural members within a Designed Handrail object using the “Convert” button on the Handrail ribbon bar. Split None connections are placed at the rail and post intersections so the user can choose which member to split.

- A Designed Handrail report is available to provide detailed member data for the Designed Handrail objects.

- To modify an assembly connection placed between members using the “Place Assembly Connection” command, the user needs to start the ‘Place Assembly Connection” command again, select the frame connection of the assembly connection you would like to modify, change the condition option on the ribbon bar to ‘Update Existing’, choose the new assembly connection type, and finish the command.

- Radial and cylindrical grid planes can be imported as reference in Sketch 2D when creating structure.

- Different part aspects (insulation, hole trace, maintenance and operation) can be brought into Sketch 2D when creating structure.

- When changing the assembly connection for a plate edge to plate face configuration to generate a chamfer, the difference between the two plate thicknesses must be greater than the bounding plate thickness plus 4mm... Below are two examples of a chamfer not being generated and for a chamfer being generated:
    - Deck plate is 20mm thick and the bounded bulkhead is 40mm thick. The difference between the two thicknesses is 20mm. The chamfer would not be generated since the difference between the two plates is not greater than 24mm (bounding deck plate plus 4mm).
    - Deck plate is 15mm thick and the bounded bulkhead is 40mm thick. The difference between the two thicknesses is 25mm. The chamfer would be generated since the difference between the two plates is greater than 19mm (bounding deck plate plus 4mm).

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Systems and Specifications

- When specifications are allowed for a parent system, users can easily allow the same specifications as to a child system by clicking the "Add All" button in the Define Allowed Specifications dialog while the child system is selected in the systems hierarchy.

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Translators

- Equipment Import now creates equipment items as GType shapes when an equipment item fails to import as a Smart 3D equivalent Shape, Nozzle or Equipment Component. This also occurs when the XML Equipment item is a PDS Dumb Graphic or a PDS User Projected Shape. To use this feature, the attribute, @ImportAsGTypeShape = “true”, must be included in the XML file. Users can export this attribute to the XML file by adding @ImportAsGTypeShape = “true”, to the EquipmentExportConfig.txt file located in the [*PDS_Installation_Directory*]\PDSExportToSP3D\config, under the /Equipment/User Attributes section.

- There are known limitations that require manual modifications after Smart 3D Import. In most cases, To Do List entries are created during Smart 3D Import.

  - Components do not get imported if the equivalent parts are not available in the catalog.
  - In a closed fitting to fitting connection scenario, a piping network distorts if the port to port dimensions of the parts do not match with the coordinate values in the XML file.

- Because of issues on the XMpLant Export side, Equipment data XML files exported from PDMS and PDS have opposite nozzle orientation. The "ShapesAndNozzlesAdditionalAttributesMap.xml" mapping file in the *[Smart_3D_Installation_Directory]**Translators**ConfigurationFiles**XMpLantSP3DMaps* *Equipment* folder needs to be modified by swapping uncommented lines and commented lines for the "Adjustment Rotation Matrix" for a "PipingStraightNozzle". The user will need to comment out the line that is currently uncommented and uncomment the line below it.

- To export a PEHVAC project that is integrated with a PDS project, the ProjList.txt file must be present in the parent folder of the PDS HVAC models folder. Additionally, if the name of PEHVAC project differs from the name of PDS project, specify PDS project name at the beginning and PEHVAC project name at the end in the ProjList.txt file.

- The catalog data must be the same for both Smart 3D and PDS for the export to yield correct results.

- The user needs to resolve all the To-Do-Lists in S3D before exporting to PDS. For example, Pipe bends modeled incorrectly in Smart 3D may result in irrelevant graphics when imported. If the user ignores to resolve all the To-Do-Lists in Smart 3D, the user may see unexpected results in PDS.

- User should use only PDS mixed projects or metric projects with metric system of units as model units for the current version of the software.

- For equipment nozzles, NPD data will be exported as per model settings using the NPD Equivalence rule which must be same on both S3D and PDS.
- Export to PDS process will export all other data in SI units such as Kilogram for weight, Pascal for Pressure, and Kelvin for Temperature. It is recommended that the physical units on PDS projects are set to SI units to avoid conflicts between the exported property value and its unit type.

- The Export to PDS process will export insulation thickness data in NPD system of units only and the insulation thickness will always follow model units for Equipment. Care must be taken while creating models for piping discipline such that the insulation thickness follows NPD system of units.

- MicroStation doesn't support the .dgn files which exceed more than 32 MB. User has to take care not to exceed the limits especially in the case of SAT files. Split the SAT file shapes in this case.

- Smart 3D exports Piping data into APL files and these files can be imported into PDS. The below points need to be considered while exporting Piping data:
  - The number of pipelines created in PDS by the export process may be higher than the number of pipe runs exported from Smart 3D.
  - Stock parts in Smart 3D which do not meet the minimum pipe length criteria in PDS, result in a piping placement error after importing.
  - The graphics for some components, such as fire hydrants, fire monitors, and so on, may be different in PDS than they were in Smart 3D.

- When the user exports spectacle blinds to PDS without setting the base part of the spectacle blind feature to either spectacle blind is in open position or spectacle blind is in closed position, the software creates an open blind in PDS by default.
- Spacers in the flanged specification are exported to PDS as 4CYL specialty items. The length of the spacer or distance piece must be mapped to the dimension DIM1 on this specialty.
- Distance pieces are not supported in the current version of the software.
- Piping Components are exported based on commodity class type.
- Service limits must be defined on the pipe run before exporting to PDS.
- The dimensional data will be exported in metric units (MMs) only for instruments and specialties.
- Pipelines with both imperial and metric specification should not be exported to the same APL file.
- The Sequence number attribute present on pipe runs in Smart 3D will not be exported into PDS.

- For plate systems with names containing invalid Microsoft file name characters, the external file name created will have name format of <<OIDoftheObject>>.<<FileTypeExtension>>.

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Weight and Center of Gravity

- Weight and Center of Gravity functionality is under the Tools menu. It is available under all tasks except Catalog, Civil, Drawings & Reports, Structural Analysis and System & Specs task.

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New Features in 2016

Details of new features in Smart 3D Version 2016 are described in the **Smart 3D Release Bulletin** located in the Help folder on the product DVD and installed with the product (<*Smart_3D_Installation_Directory*>*Documentation**Help**ReleaseBulletin.pdf*).

New Fixes in 2016

Setup

TR260244
PDS Translators and Tribon Interface are now observing user-defined installation path, rather than the default C:\Program Files (x86)\Smart3D.

TR261935
A confirmation message is now displayed when uninstalling Smart 3D.

TR275086
Users must now accept Smart 3D software license agreement during silent installation.

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Catalog Data / Reference Data

TR191299, TR266895
If the calculated Cap Screw length is not available in the Preferred Bolt Length table, the bolt length calculated is incorrect. It is fixed such that a To Do List is generated for incorrect Cap Screw lengths.

TR204597
Users can now delete Fireproofing specifications.

TR213934
Users are unable to change the Short code for Branch Table in Catalog Task. This is now fixed.

TR215176, TR270214
Water COG X value is now correctly populated in catalog task. **(Also implemented in 10.01.15.0070)**

TR222620
Automated Selection of Parts at Spec Break Rule will now work for Versionable Specifications.

TR223921
When the Plate Flange Setback Distance Rule is defined with different NPD units, the correct setback value is now selected based on NPD and NPD units.

TR224408
When the user modifies a turn feature to a miter, the Bend Radius Multiplier value is now shown correctly on the property page.

TR232893
Materials Management Ident property is now available for all connection parts.

TR251490
Users can now bulkload Insulation Thickness rule successfully when Pipe Spec Version Management is enabled.

TR254949
In Oracle datasets, Insulation Thickness values are now sorted in ascending order on the properties dialog.

TR255146
Problems during the handling of bulkloaded custom codelisted attributes have been mitigated by the enforcement of a data type check, which will only allow Long and Short attribute types in the bulkload.

TR258592, TR264348, TR264349, TR264350
Users are now able to filter equipment based on a property in the Catalog browser in grid view. **(Also implemented in 09.01.30.0099, 10.00.73.0063, 10.01.15.0065)**

TR262925
Users are now restricted from bulkloading Specifications rules when the user does not have permissions on the required Catalog Permission group.

TR265144, TR266360, TR256440
Verify Consistency Report is now able to populate Bolt Length Calculations data. **(Also implemented in 10.00.73.0065, 10.01.15.0067)**

TR270401, TR276926, TR276927, TR276928
The performance of “Procurement Client Commodity Code” Label is now improved. **(Also implemented in 09.01.30.0113, 10.00.73.0077, 10.01.15.0078)**

TR273174, TR280159, TR280409, TR280410
Modification of Equipment Insulation Material data will no longer affect Piping Insulation Material data. **(Also implemented in 09.01.30.0116, 10.00.73.0078, 10.01.15.0079)**

TR274702, TR275515
Bulkload now correctly handles Piping Specifications having zeros after decimals in their Specification names. **(Also implemented in 10.01.15.0077)**

TR276378
“No Gasket Found” errors appearing for components using Integral Gasket requirements when connected to equipment nozzles have been addressed.

TR280473, TR282810, TR283098
When the “Minimum Length Check” rule is enabled on the Pipe bend properties dialog, an invalid To Do List entry is created and if the user further modifies the bend type, it is not persisted. This is now fixed. **(Also implemented in 10.00.73.0082, 10.01.15.0082)**

TR282198, TR283167, TR283382, TR283383
“Smart3D Compare Catalogs” utility now successfully compares the properties of Hangers and Supports objects. **(Also implemented in 09.01.30.0119, 10.00.73.0082, 10.01.15.0082)**

TR285820, TR286410, TR286928
When the Bolt length lookup rule is enabled, Bolt commodity code substitution rule is now working. **(Also implemented in 10.00.73.0085, 10.01.15.0085)**

TR286363
The bulkload .lst file now lists the loading requirements for Structure datafiles.

TR287260
The bulkload tool will now delete “A/M” after bulkload, even for large workbooks.

TR288101, TR290179
A few properties like Part Data basis, Geometry Type are not displayed correctly for Piping Instruments and Specialties in the Catalog task.

TR288799, TR290378
Bulkload fails in specific Oracle datasets if the user loads different workbooks in multiple iterations without reopening the utility.

TR289880
A few labels failed in a specific dataset after upgrading to 2014 R1, as the relationship between CPPipeStock Class and IJUAManufacturerData is not available.

TR290963
When the User Account Control setting is enabled to "level 3" and the "virtual store" setting is disabled in a Windows Operating system, the Bulkload Utility displays an error in specific machines.

TR293228
The “SelectionChanged” event is now triggered only once for the “CatalogComboBoxView” control after clicking OK in the Catalog Browser custom command dialog.

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Civil

TR283180
Trench Boundaries will no longer overlap when routing a new trench run from an existing trench run.

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Common

TR156325
Distance can be measured between two points on a Reference 3D Model attached from CADWorx.

TR174478
Tripping Stiffeners on extruded plates now can be copied using Copy Symmetry and Copy Similar commands.

TR188773
If user selects the "Modify" command and immediately selects the "Delete" command, without select "reject" or "FINISH", an error occurs and the Copy by Family command aborts.

TR189080, TR217180, TR259026
Right-click menu is now displayed in the Workspace Explorer after using the Measure command. **(Also implemented in 10.00.73.0059, 10.01.15.0062)**

TR190277
PointAtMinDistance and Dimension constraints now can be copied using Copy Symmetry and Copy Similar commands.

TR205264
Control Point now shows correct coordinates when active coordinate system is other than Global. Webcut parameters are also showing correct values after multiple changes of the end cut type in the Assembly Connections properties page.

TR211476, TR277496
The Active View Control window stays up even after the user switches tasks. **(Also implemented in 10.01.15.0079)**

TR212688
Mutual circular bounded structure using tripping stiffeners & sketched plates now can be copied using Copy Symmetry and Copy Similar commands.

TR227226
The cause for the display of an Application Error message after the selection of Reference 3D member parts using the delivered “Reference 3D Member Part” filter has been fixed.

TR248981
“Exception in Filter” error message when opening browser for resolving cross-overs has been fixed in Copy Across Model command.

TR249560
QuickPick now shows the Point on Surface glyph when moving piping, and users should be able to select the top and bottom of steel as reference points.

TR249681
Curved member with full circular path is copied with correct orientation. This supports using Copy by Family and Model Data Reuse.

TR252511
The cause for crashes related to closing the Repository Browser in Smart 3D Marine environments has been addressed.

TR256936, TR260790
Full Layer names are now displayed on the Levels/Layers tab of Reference 3D property form in the Workspace Explorer. **(Also implemented in 10.01.15.0064)**

TR257400
If a saved session file has its render mode changed from “Shaded with Hardware Enhanced Edges” to “Smooth Shaded” in a marine workspace, the view axis will not change from X-Y-Z to E-N-El after closing and re-opening the session file.

TR258464
Property filters created using a comma as the decimal delimiter now works.

TR259318
Updated Create Offset Curve process to handle more geometry cases. **(Also implemented in 09.01.30.0096)**

TR259367
Memory issues on earlier versions of the Automation toolkit have been fixed in order to avoid crashes.

TR259744
To Do List entries in the model are now filtered by the current workspace.

TR262667
The Metadata Browser tool has been cleaned to stop displaying attributes that are not accessible for the user to work with.

TR263865
Users will not be able to modify filter properties if the filter is in Approved status.

TR264724, TR266650
Attributes for Hanger assemblies will now correctly return property values for parent\child values. **(Also implemented in 10.01.15.0079)**

TR267116
User can now measure the distance between two points on Reference 3D models attached from CADWorx.

TR267451
Future issues with attribute name handling have been mitigated by the enforcement of a 30 character limit in both SQL and Oracle environments.

TR268234
Pinpoint values are now retained when moving cross section segment of volume by path.

TR269608, TR269996, TR269997, TR269998
Incorrect To Do List records are not generated in a Global Workshare configuration (GWC) when running the Synchronize Model with Catalog command without the out of date option selected. **(Also implemented in 09.00.30.0104, 10.00.73.0070, 10.01.15.0069)**

TR270230
Sketch features will be copied or modifed using Copy Symmetry and Copy Similar commands even if the plate has slots.

TR270717, TR272712
Define Workspace workflow works as designed for any R3D objects.
(Also implemented in 10.01.15.0072)

TR272955
Update Database Integrity scripts now run successfully in the model.

TR273119, TR276057
Performance of Copy By Family command in cross-over cases has been improved. (**Also implemented in 10.01.15.0067**)

TR273431
The cause behind failed transaction error messages after switching between models in the Define Workspace window has been fixed.

TR274561
Batch update of Reference 3D projects submitted to a remote server machine is now properly updating projects.

TR275344
Users can now place handrails and walls around trench or slab edges created from openings.

TR276480
Smart 3D does not close when selecting the Clipboard window option in the paste dialog.

TR280751
Sessions no longer crash when selecting the Show Clipboard window while pasting copied pipe runs.

TR281859
Get SymbolShare now works in the Standalone App if Database Wizard wasn't used to set Site.

TR282292
Error message is not displayed when a session file is refreshed.

TR284213, TR284214
Enhanced Database Integrity scripts prevent handrail deletion from the model. This is observed only when placing the best practice stair using the symbol based method. **(Also implemented in 10.01.15.0083)**

TR285860
Tooltip for ruler is now displayed correctly.

TR286072
S3D Host crash during the "Update out-of-date occurrences" phase in the Synchronize Model with Catalog process with structural detailing objects has been fixed.

TR290736
Copy Symmetry family is created even with unequal number of frames on positive and negative side of co-ordinate system.

TR290963
Bulkload Utility now passes all certification tests with UAC set to "level3" and "virtual store" disabled.

TR291480
Errors thrown during the deletion of WBS Items and piping objects related to WBS items have been fixed.

TR294312
Errors encountered during the modification of Drawing Volumes by users in Global Workshare with access to the objects have been addressed.

TR296524
When using Copy Similar to copy a bulkhead with profiles created from Geometric Constructions, some of the profiles did not copy. The following profiles now copy:

- Profile created by GCPointsAlongCurve and GCCurveByPointsOnSurface methods
- Profile created by GCVerticesFromCurveOrSurf and GCCurveBy2PointsOnSurface methods

TR297192
The selection in Model Data Reuse of a filter that includes a volume in its definition is no longer rejected when the "Copy volumes in range" option is not selected.

TR299864
Plates are now getting mapped while running "Find and Correlate" command when more than one root plate with in the same range are existing in the source section.

TR301021
Clip by Volume Command can now redefine Depth Points.

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Common Route

TR80505
Automatic reconnection works when moving a straight piping feature onto another to form a perpendicular connection with olet or tee branching.

TR156696
The repair script executed via the Clean Database command now sets route entries to Resolved if the objects were successfully updated.

TR209659
A value for BendRadiusMultiplier is no longer displayed for elbows where the BendRadiusMultiplier is not defined in the part data.

TR233124
Material description labels for connection parts are now displayed correctly in the properties page.

TR266928, TR269483, TR269485, TR269486
Route now avoids precision beyond the defined tolerance for system generated features. (**Also implemented in 09.01.30.0104, 10.00.73.0068, 10.01.15.0069**)

TR273909, TR274635, TR274640
Plane by Three Points works after the implementation of the Remember Working Plane functionality. (**Also implemented in 10.00.73.0076, 10.01.15.0077**)

TR289783
Modifications to the Fabrication Requirement and Fabrication Type of a gasket at a specification break, now holds the modified values.

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Compartmentation

TR271297
Performance has been improved for the creation of compartments when type of compartment is from the compartments class under the Compartment Definitions.

TR272184
Performance has been improved for Compartment Painting Area Report on large databases.

TR272226
Compartment Painting Area Report now succeeds when leaf plate of a boundary is set to inactive. It also does not fail when a lapped plate is created on a leaf plate of a boundary.

TR286531
All parts falling inside compartment volume are now reported in Compartment Paint Area Report.

TR289546
When using the Create Compartments Automatically command, all of the compartments created are now created correctly.

TR295285
When creating a volume by window, the user selects a folder for the space \ volume to be placed. After the initial generate, the volume is now placed in the desired folder.

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SmartSketch Drawing Editor

TR150349
The underline label border no longer changes to the over line label border when the label's location changes to -y position relative to the leader's start point.

TR177689
The 'TAB' key now advances to the next step in Sketch2D, when invoked.

TR185191
Speed of SmartSketch is considerably improved with respect to the following scenarios.

- View Updates are responsive without flickering and only a single, final display appears.
- Large files can be opened with better performance and speed than previous versions.
- No more reading of resource files such as 'Styles' while loading from 'Open File'.

TR186066
Save as DGN no longer creates multiple files with different scales. Users can now set the option 'Merge References = 1' similar to the AutoCAD merging in the translator's configuration file 'itmstn.ini' located in <*Smart_3D_Installation_Directory*>*Common2D**Rad2d**Bin* folder.

TR186821
Changes have been made so that the elements with segmented line style (i.e., with the combination of hidden & visible line styles depending on their z order) is exported as a single group containing the disjoint entity segments. This helps in selecting the entire element as a single entity for any manipulations.

TR194070, TR213632
Changes have been made to retain the z order from SmartSketch Drawing Editor. This improves the display when exported to AutoCAD which in turn results in WYSIWYG.

TR198050, TR259256, TR259257, TR259258
Users can now use the Pinpoint relative tracking with SmartSketch Drawing Editor modification commands such as Move/Copy. **(Also implemented in 09.01.30.0097, 10.00.73.0059, 10.01.15.0062)**

TR235524
Dimension elements in SmartSketch Drawing Editor retain their graphics, especially the arrow heads even after you drop them to dumb graphics and export to AutoCAD.

TR238622, TR266185, TR266239, TR266240
Users can convert using a customized RGB color from SmartSketch to AutoCAD to the nearest possible match. **(Also implemented in 09.01.30.0101, 10.00.73.0065, 10.01.15.0067)**

TR242621
Changes have been made to prevent SmartFrame border (which is switched off by unchecking the 'Show Border' on drawing view properties) to appear in the exported DWG or DXF file.

TR242757
Enabling PrintManager (from the SmarkSketch Support Utilities Pack) no longer restricts in saving the border graphics (background sheets) to PDF.

TR246234
Symbol Text no longer flips its orientation when exported to MicroStation.

TR247826
Changes have been made to retain the z order from SmartSketch Drawing Editor. This improves the display when exported to MicroStation which in turn results in WYSIWYG.

TR251483
Improvements have been made to handle the closed curves which export the fill patterns of customer logos to MicroStation to achieve WYSIWYG.

TR251990, TR260358, TR260359
The text which is mapped using TrueType Fonts in MSTN font.rsc file, displays correctly at all times i.e., Zoom-In or Zoom-Out. Improvements have been made to 'Stroke Test = 1' option which handles the exported TEXT to display properly. **(Also implemented in 10.00.73.0060, 10.01.15.0064)**

TR251599, TR260800, TR260801
Fractional dimensions now retain the dimensional units of ft-in when exported to MicroStation. **(Also implemented in 10.00.73.0061, 10.01.15.0064)**

TR262401, TR265527, TR265528, TR265587, TR270474
The text which is placed with Text Alignment of Left Top translates correctly to MicroStation V7 or V8 formats. **(Also implemented in 09.01.30.0100, 10.00.73.0064, 10.01.15.0067)**

TR267573, TR284441
Improvements have been made to handle large files which are failing to convert into ACAD .dxf/.dwg formats. When large files are getting translated, the translation completes successfully whereas the binding (to convert into a single file merging all the references using Merge References = 1 option) operation fails due to memory. The new changes will now prevent binding if there are memory limitations. **(Also implemented in 10.01.15.0084)**

TR267953
Installing SmartPlant P&ID no longer breaks SmartSketch Drawing Editor. Users can install SmartPlant P&ID after SmartSketch Drawing Editor.

TR269066
Custom Line Styles with asymmetric caps can be now converted to AutoCAD without any distortions.

TR269667, TR282786, TR284651, TR284652
Text border for Roman D fonts translates to AutoCAD with proper graphics. **(Also implemented in 09.01.30.0120, 10.00.73.0085, 10.01.15.0084)**

TR275906, TR280636, TR282124
While exporting fill patterns, increase the itmstn.ini setting 'MAX HATCH STROKES' to translate properly to MicroStation. Changes have been made to notify users with a message box if the hatch strokes exceed the maximum value specified in the itmstn.ini file. **(Also implemented in 10.01.15.0082, 10.01.15.0085)**

TR278623, TR282238
Changes have been made to export the correct color tables to MicroStation V8 formats. **(Also implemented in 10.01.15.0084)**

TR280271
'Export Graphic Groups = 1' can be used in the options file itacad.ini and itmstn.ini to properly export objects without exploding. This is applicable for the elements which are contained within groups.

TR287112
Users can map True Type Fonts or Intergraph native fonts to MicroStation native fonts Ex: Arial = Engineering, Intergraph ANSI = Engineering. MicroStation font's resource file has mapping between Native Fonts Vs Font ID, so it will internally manipulate and translate the fonts to MicroStation V8 format.

TR292619, TR294853
When translating an SHA file to V8 DGN file, the dot terminators on leader lines no longer translate to hollow arrow terminators. **(Also implemented in 10.01.15.0091)**

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Drawings

TR75621
Copy/Paste of Drawings by Query Manager component now preserves the 'Filter definition' and 'Package details'.

TR94887, TR262942, TR266547
Users can now handle revisions/issues either with respect to tool (Smart 3D) or with respect to SPF in an integrated environment. 'ConfigureDrawingProjectSettings.exe' under ProjectMgmt\Tools\Bin helps in this configuration. See the documentation for more details. **(Also implemented in 10.00.73.0066, 10.01.15.0068)**

TR95175
Drawing quality has improved when processing the HVAC and Cable Trays.

TR114908, TR257857, TR257863, TR257864
Users can now generate SPRDirect and 3D Model Data files using SQL filters. **(Also implemented in 09.01.30.0097, 10.00.73.0060,10.01.15.0061)**

TR127682
Users can now create drawings without opening a new session if they have multiple plants. They can switch from one plant to another and proceed with creating drawings without any errors.

TR143700
Snapshot settings are now remembered in the session. All settings, with the exception of View Name will be automatically populated (from previous usage) when running the snapshot command. View Name will not be populated if the Naming Rule is set to 'User Defined'.

TR144035
'Clipping' option on View Style works for all the curves such as 'Grid Lines'.

TR144233, TR296913
Batch update has a new option "Full Update", which works for all the drawing types. **(Also implemented in 10.01.15.0095)**

TR162455
Delivered label 'Electrical Instrument Plan_Instrument_Name' now works on Oracle databases as well.

TR164125, TR275741
Batch processing now supports new option 'Save As' in addition to 'Refresh' and 'Update' for all the drawing types. This allows the users to save the drawings to the physical location after update using batch. **(Also implemented in 10.01.15.0080)**

TR178478
Parametric symbols defined for Valves gets oriented on drawing outputs with respect to their placement in the model.

TR196182, 277150
Users can change the permission groups of multiple drawings selected from the Drawings and Reports environment in a Global Workshare configuration. This behaves in the same way as any other object types such as Piping Components / Equipment where users can select multiple objects and change their permission group. **(Also implemented in 10.01.15.0080)**

TR196898
View remains in locked state even after editing the drawing view properties.

TR221763, TR283316
Generating a preview for a steel order reference plane drawing is now supported. **(Also implemented in 10.01.15.0084)**

TR231213
Enhanced label services will now report the 'Reference Bubble labels' in all cases.

TR242227, TR275642
All Tangent edges are now configured with an additional line style 'Tangent Edge Visible Line Style' in Graphic Rules which is by default set to 'Transparent'. **(Also implemented in 10.01.15.0077)**

TR245536
Enhanced geometric analyzers for 'DrawingGAMarginOnly' and 'DrawingGAMaringOnlyWithOccludedGraphics' helps in getting the anchor dimensions for all combinations.

TR252961, TR258827, TR258828, TR258829
Pile Layouts modeled using Design Solid shapes will now get labeled on drawings even though the shapes are clipped by the volume. **(Also implemented in 09.01.30.0095, 10.00.73.0059, 10.01.15.0062)**

TR255738
Pasted Drawings now inherit properties from the parent component similar to the way create drawing functions.

TR256948, TR259600, TR259601, TR259602, TR259603
Enhanced Drawing VHL processing eliminates missing graphics resulting from Reference 3D references. **(Also implemented in 09.00.71.0103, 09.01.30.0097, 10.00.73.0059, 10.01.15.0062)**

TR257297, TR267176, TR267417
Copy/Paste of source folder will now copy/paste its child components in the Drawings and Reports environment. **(Also implemented in 10.00.73.0066, 10.01.15.0068)**

TR258306, TR266174, TR266493, TR266495
Section surface is now created at the center of the web, or on the web surface, for standard members in scantling drawings. **(Also implemented in 09.01.30.0101, 10.00.73.0066, 10.01.15.0067)**

TR258851, TR262086
Enhanced 'PenPlatePointGenerator' module avoids duplicate radial dimensions when the center point and radius of the arc objects are same. **(Also implemented in 10.01.15.0065)**

TR258862, TR263804, TR263805, TR263806
New Geometric Analyzer module 'DrawingGAMarginOnlyWithOccludedGraphics' helps in handling the retention of dimensions for occluded objects on drawings. **(Also implemented in 09.01.30.0099, 10.00.73.0063, 10.01.15.0065)**

TR259194, TR266916, TR266939, TR266940
Drawing views/attributes gets stripped off from the select set when creating symbols. This avoids drawing corruptions. **(Also implemented in 09.01.30.0102, 10.00.73.0066, 10.01.15.0068)**

TR260654, TR271505
Improved Math API's to skip bad geometries during drawing updates in order to prevent unhandled exception errors. **(Also implemented in 10.01.15.0072)**

TR260960, TR268907, TR268908
Enhanced behavior of handling Styles.sha prevents issues with saving the drawings after update. If the Styles.sha is saved with 'Background Sheets' active, then software automatically corrects it by saving it to 'Working sheet', this prevents the issues with saving the drawings after updating them. **(Also implemented in 10.00.73.0069, 10.01.15.0070)**

TR261118, TR263343, TR263344, TR263345
Enhanced 'Make Drawable' wrapper preparation rule eliminates objects which implement 'IJDCoordinateSystem' from causing drawing VHL failures. **(Also implemented in 09.01.30.0100, 10.00.73.0063, 10.01.15.0065)**

TR261469
'Inside Leader' placement option works even after invoking the label placement.

TR261508, TR266029, TR269924
Hidden profiles that are defined in the view style to not be drawn, are no longer getting drawn graphically in composed drawings. **(Also implemented in 10.00.73.0066, 10.01.15.0068)**

TR261750, TR265468, TR265469, TR265470
View name and view labels no longer fail to update, when the primary input object is changed, upon full update of view. **(Also implemented in 09.01.30.0103, 10.00.73.0067, 10.01.15.0068)**

TR261861, TR266445, TR266446, TR266447
Flange direction is no longer drawn opposite of correct orientation when the load point is changed to centered or inner edge. **(Also implemented in 09.01.30.0101, 10.00.73.0066, 10.01.15.0067)**

TR262032, TR266503, TR266504, TR266505
Profile length for bulb profiles is no longer short 2mm when 'Drawing Rule' is set to 'Draw Profile Web left surface'. **(Also implemented in 09.01.30.0103, 10.00.73.0067, 10.01.15.0068)**

TR262241
Multiple tool bars are no longer created when section and detail views are selected and the middle mouse button is used to zoom.

TR262897
Enhanced 'Design Support' navigation rule now includes both supporting and supported objects.

TR263288, TR264158
Changes made in 'allowAcuteBreaklineAngle' enhances the leader placement options. **(Also implemented in 10.01.15.0065)**

TR265774, TR283470
New option 'ScaleFactor' is now available to define the required scale factor for flow arrows on Piping GA Drawings. **(Also implemented in 10.01.15.0082)**

TR268589, TR275052
Improved Math API's to draw missing graphics for certain shapes under a design equipment. **(Also implemented in 10.01.15.0077)**

TR269075, TR272188
Improved Math API's to create boundaries for R3D objects which in turn results in complete VHL output on drawings. **(Also implemented in 10.01.15.0071)**

TR269131, TR270203
Drawing volume no longer increases when an object is drawn with clipping 'off' in the view style. **(Also implemented in 10.01.15.0070)**

TR270318, TR271147, TR271328, TR271478
Enhanced Batch Update process to prevent errors with incorrect Smart Frame ID's. **(Also implemented in 09.01.30.0106, 10.00.73.0070, 10.01.15.0071)**

TR272701, TR274277
Embedded SQL report in drawings will no longer give blank output when updated using 64-bit Batch. **(Also implemented in 10.01.15.0077)**

TR273605, TR273892
Improved Math API's to handle missing pipe components on drawings. **(Also implemented in 10.01.15.0073)**

TR273787, TR274133, TR274270
Steel Order by Reference Plane drawings no longer fail to update in environments with a delimiter being anything other than a decimal point. **(Also implemented in 10.00.73.0075, 10.01.15.0076)**

TR274194, TR275636, TR275637, TR275639
Improved mechanism for handling clipped objects to prevent missing dimensions after updates. **(Also implemented in 09.01.30.0114, 10.00.73.0077, 10.01.15.0078)**

TR274365, TR275742
Users can modify the drawing sheet naming rule after the initial creation. Furthermore, it retains the current Name Rule and all of its properties after these modifications. **(Also implemented in 10.01.15.0081)**

TR274613, TR275631, TR275633
Section and Detail views that have been converted to independent views no longer fail to be updated with a "Full Update". **(Also implemented in 10.00.73.0075, 10.01.15.0076)**

TR274654, TR275771, TR275772
For Drawings by Rule, manual changes to view names are now maintained after update. **(Also implemented in 10.00.73.0075, 10.01.15.0076)**

TR275421, TR277125, TR277126
Profiles using the Landing Curve with End Cuts graphic rule now use the correct line style. **(Also implemented in 10.00.73.0082, 10.01.15.0082)**

TR275861, TR278654
Clipping settings now only apply to view extents, the full projection in the view direction of the object is shown. **(Also implemented in 10.01.15.0079)**

TR277031, TR280027
Volume growth values now function properly for oriented bounding box volumes. **(Also implemented in 10.01.15.0080)**

TR277680, TR281279, TR281364
New Drawing Wrapper 'MakeDrawableAssembly' now available to handle assembly objects. This helps in properly drawing all the child components under an Assembly without defining additional rows in the view style. **(Also implemented in 10.00.73.0081, 10.01.15.0081)**

TR280361
Design Equipment which consists of design solids are now getting drawn. This allows the equipment Shapes under Design Solids to be labeled if modeled as supporting structure in Hangers DBQ Drawings.

TR281119, TR285039
Converted delivered labels which are in SQL to COM, which in turn enhanced the performance of 3D Model Data updates. **(Also implemented in 10.01.15.0090)**

TR282096
Profile Length for bulb profiles is no longer short by 2mm when the Graphic Rule Type being used is 'Visible/Hidden Edges (VHL)'.

TR282361
The Oracle query has been enhanced to handle gathering phase of objects while updating the drawings. This improves the drawing update time.

TR283040
'DrawingLinearLeader' leader module, no longer ignores the breakline setting.

TR283521, TR284385
Improved Math API's to prevent unhandled exceptions caused from moving section views from one drawing to another. **(Also implemented in 10.01.15.0083)**

TR283860
Hidden profiles are no longer shown as visible when the cutting surface by intersection graphic rule is used.

TR284459
"Tests on Parts" orientation check for features is now applying chamfer labels to chamfer features on target plates.

TR284658
Composed Drawings smart frame origin now matches with its Range values (X-Minimum and Y-Minimum) from the model.

TR285550, TR285786
When Drawing by Rule views are copied and then pasted using a different orientation rule, the Looking and Up direction properties function properly. **(Also implemented in 10.01.15.0084)**

TR285572
Enhanced handling of getting the symbol origin for some piping components, such as Butterfly Valves, now prevents the incorrect orientations of symbols when used as 'Replace Object with Symbol' option in drawings.

TR285882
Profile end cut features no longer fail to re-symbolize when a profile sketch drawing fails.

TR285913
'Update and Publish' behavior has been enhanced to handle both Update and Publish in separate processes. This allows the users to move on to their next task without waiting.

TR285932
Non-target profiles bounded to target profiles which are drawn "resymbolize by ruleset" no longer have visibility errors.

TR288642
When the user has a Drawing by Rule view which is linked to a ruleset view style, and then the view style is either renamed or deleted in the symbol share, selecting a new view style through the view properties no longer produces errors.

TR288884
When updating an Assembly Drawing using the RuleSetStyles/Assembly view style, the drawing will no longer stall during the annotation phase and cause S3DTaskHost to crash.

TR290116, TR291321
Enhanced drawing update process which deletes the unwanted objects that can't be deleted manually. This prevents from drawing update failures. **(Also implemented in 10.01.15.0090)**

TR290592
'Permission group' on drawing volumes remain in sync with the drawing template until unless the drawing is created. Once the drawing is created using a different Permission Group, the volumes Permission Group changes accordingly.

TR291033
When View Scale is set as "Fit to Scale", End cuts are now resized when drawing is updated for first time even if we set "NORMALIZATION" as true in the profile end cut XML.

TR294753
The Composed drawing views honor the orientation of the volume after a volume rotation when the 'View Direction Coordinate System' is set to 'Use Object CS'.

TR294915
Enhanced 'GridLinesDrawingWrapperEntity' wrapper supports the sloped grid planes to appear in drawings.

TR295126, TR169376, TR301445
The leader now honors and gets repositioned when its corresponding object is moved in the model which are replaced with symbols using 'Replace Object with Symbols' graphic rules. **(Also implemented in 10.01.15.0096)**

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Electrical

TR223346
Cabletray barriers are no longer distorted when routing from a vertical outside bend.

TR224308
If the user does not enter values in any of the inputs in the Cable properties dialog, the user is forced to re-open the dialog and key-in all the cable properties again. This issue is now fixed.

TR240324
Cableway splits are no longer lost when the user modifies the cableway of some cabletray parts.

TR247423
The default bend radius defined at the specification level is now honored when routing a cabletray.

TR251085
A valid message, "No Part found in Catalog Data" message is now displayed if a user tries to route a conduit for which conduit stock part is not defined.

TR261819, TR268755, TR270604
The route status of a cable passing through the branch feature of a Tee is no longer changed to "Incomplete" when the branch feature is replaced. **(Also implemented in 10.00.73.0071, 10.01.15.0073)**

TR262125
Reportable barrier parts are now generated for components inserted manually.

TR264070
The status of cable routed through a cabletray network that has cableways is no longer displayed as "Incomplete" if the user splits the cabletray and replaces the part with a cableway.

TR275303
Enclosed conduit runs of a duct bank can now be routed with slope for slope values less than 9mm/1m.

TR281782
Cable route status is now displayed as “Incomplete” and “The entry/exit for the cable is not specified or missing” To Do List is created if a split is inserted on the cabletray close to the point where cable is terminated in a closed-loop cabletray network.

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Equipment and Furnishings

TR190767, TR262160
Custom Name Rules are now shown in the properties page before placing equipment or equipment components. **(Also implemented in 10.01.15.0065)**

TR230432
Extra lines are no longer shown on equipment when viewed orthogonally using the Shaded with Hardware Enhanced Edges render mode.

TR237466
Nozzles do not get deleted when its parent shape is moved under a Design Solid.

TR252493, TR261142, TR261612
Users can now select a nozzle axis when creating a Parallel and Mate to Tangent relationship. **(Also implemented in 10.00.73.0061, 10.01.15.0064)**

TR254165, TR261729, TR261793, TR261794
The method to get and set the hole diameter of foundation port using Smart3D .Net API is now available. **(Also implemented in 09.01.30.0098, 10.00.73.0061, 10.01.15.0064)**

TR261666, TR263905, TR263906
‘Eccentric Pyramid’ shape can now be placed under a Design Solid. **(Also implemented in 10.00.73.0063, 10.01.15.0065)**

TR262429
A Foundation Port placed using “Position by Point” placement type now moves along with its parent object.

TR263615, TR263907, TR263908, TR265850, TR265851
In a specific customer dataset, Interference detection report an equipment as bad part as its child shapes were in To-Do List. **(Also implemented in 09.00.71.0103, 09.01.30.0099, 10.00.73.0063, 10.01.15.0068)**

TR267289, TR276814
Users can now place equipment when the design support is a child of another equipment. **(Also implemented in 10.01.15.0078)**

TR271262, TR272411, TR273053, TR273054
The CoG values for Equipment and Equipment Components can now be set using Smart3D .NET API. **(Also implemented in 09.01.30.0107, 10.00.73.0071, 10.01.15.0072)**

TR287663
Microstation .dgn files can now be imported as a shape even without defining the master units in the file.

TR297135
Graphics for piping elbow nozzle symbol will display the graphics properly. With this fix, for the piping female Elbow Nozzle with FTE/SWE end, the connection port will come inside the nozzle.

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Geometry Analysis and Repair

TR281222
Knuckle lines on Port and Starboard sides of the hull are now continuous after exporting the hull.sat file to the Molded Forms task by Import Plate System command.

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Grids

TR220499
When creating a new coordinate system (CS) with the Grid Wizard, the user can now use a CS that does not have any axes and that the user does not have write permissions.

TR233166
Rulers behave consistently when the user manipulates the model view in the workspace. Rulers no longer disappear or show gaps in ruler marks due to manipulating the model view.

TR243665
S3D Session no longer hangs when trying to view the grid plane properties.

TR263185
Grid systems can now be exported with a user-defined name containing special characters.

TR263439
Ship grid type no longer appears as a grid type when using Grid Wizard in an MHE model.

TR268490
Grid Wizard now works if an unnamed coordinate systems exist in a model.

TR 295283
User now has the ability to increase negative grid extension value.

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Hangers and Supports

TR241808, TR260369, TR261487, TR261488
Supports can now be placed using footings as supporting structure. **(Also implemented in 09.01.30.0098, 10.00.73.0061, 10.01.15.0064)**

TR256984, TR282428
Users will be able to drop multiple standard supports at once. **(Also implemented in 10.01.15.0082)**

TR257521, TR263106, TR263107, TR263108
The total weight of a support assembly now equals the sum of all of its children components. **(Also implemented in 09.01.30.0099, 10.00.73.0063, 10.01.15.0065)**

TR260265, TR260479, TR260480, TR260481
Toggle Connections now works properly with built up and linear members. **(Also implemented in 09.01.30.0097, 10.00.73.0060, 10.01.15.0064)**

TR261215, TR261730, TR263431, TR263434
GetInsulationData method now returns the correct value for non-insulated pipes. **(Also implemented in 09.01.30.0099, 10.00.73.0063, 10.01.15.0065)**

TR262078, TR265599
Users can now modify the plate thickness and radius values of a utility curved plate. **(Also implemented in 10.01.15.0066)**

TR262663
Attributes on the assembly “Assy_FR_UC_CS” can now be modified.

TR263457, TR263898, TR263899, TR263900
Users can now set the BOM Description value for design supports using the Smart 3D .NET API. **(Also implemented in 09.01.30.0099, 10.00.73.0063, 10.01.15.0065)**

TR264394, TR266792, TR266802, TR266803
When a support note(s) is added to the support properties page, the BOM description is no longer automatically set to blank. **(Also implemented in 09.01.30.0103, 10.00.73.0067, 10.01.15.0069)**

TR266404, TR266654, TR266655
HangersAndSupportsATP task is now listed in the S3D task list upon running the “RunSP3DEUBATP.vbs” script. **(Also implemented in 10.00.73.0065, 10.01.15.0067)**

TR271081, TR276493
Supports now get aligned automatically with the Route Port when placed through Smart3D .Net API.

TR271250, TR272107
HVAC Bounding Box (BBX) hanger ports orientation is now fixed. **(Also implemented in 10.01.15.0071)**

TR274214, TR276874, TR277329
Design Supports can now be placed on miter bends. **(Also implemented in 09.01.30.0113, 10.00.73.0077)**

TR276428, TR282094, TR283321, TR283322
Users can now select an AISC-Pipe as supporting structure which belongs to a standard support. **(Also implemented in 09.01.30.0119, 10.00.73.0082**, **10.01.15.0082)**

TR276908, TR284643
A custom command for placing supports from XLS is now delivered with Smart 3D. This will work the same as RunSP3DEUBATP.vbs. **(Also implemented in 10.01.15.0084)**

TR279875, TR284744, TR286201
Users can place all the default Lisega_2010 support data using the LICAD Plugin (3rd Party App). **(Also implemented in 09.01.30.0120, 10.00.73.0085)**

TR282146, TR285617
The custom aspects defined on support components are now displayed. **(Also implemented in 10.01.15.0084)**

TR282756, TR290386
Support ports orientation and position can now be modified using HgrSupPortsOrientOnPlanarSurfaces rule. (**Also implemented in 10.00.73.0088)**

TR284055
Key point notes will not be reported on a copied support.

TR284426, TR285532
Split Root System now works with all outfitting objects. **(Also implemented in 10.01.15.0084)**

TR292895
The L sections in Unbraced Cantilever Support no longer offset from the pipe during its placement using place by point command.

TR296666
User will now be able to place the B-Line Trapeze Assembly Support on a cableway turn feature.

TR297823
Support placed on Hanger steel connects properly similar to structural steel.

TR300791
After cleaning the DBI records, “Corrupted Hanger Connections” entries are not seen anymore.

TR301011
Face Selection is now honored by profile parts.

TR302396
Euro Cross Sections can now be placed using Placing supports from XLS ATP.

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Hole Management

TR253075
Users can now fence select pipe features and multi-select hole traces without error.

TR253690, TR259556
Hole traces are now shown at the correct angle for pipes penetrating the hull. (**Also implemented on 10.00.73.0052**)

TR256424
For wall attached coaming sketch, when user sketches two perpendicular lines connected to each other and each line is end constrained with plate edges, the sketch is recognized.

TR256836
Sketched hole traces are now able to be placed on standard members.

TR258966, TR260745
Users can now select the penetration component as an outfitting part for placing a hole trace. (**Also implemented on 10.01.15.0063**)

TR259651
Hole traces are now able to be placed on a curved plate.

TR262998, TR263756, TR263757, TR263758
Users can now place sketched hole traces on Tribon curved panels without issue. (**Also implemented on 09.01.30.0099, 10.00.73.0063, 10.01.15.0065**)

TR265231, TR266034
Users can now place hole trace on Tribon plates without issue. (**Also implemented on 10.01.15.0066**)

TR267927, TR268797, TR273475
Hole trace diameter values are now correct for circular cableway coaming. (**Also implemented on 10.01.15.0073, 10.00.73.0060**)

TR272401, TR275388, TR275389, TR275390
Hole cuts are created and now are displayed correctly in Part Monitor outputs. (**Also implemented in 09.01.30.0112, 10.00.73.0075, 10.01.15.0076**)

TR272472
Users can now place hole traces using the Multiple Hole command without issue.

TR275845
New hole cuts using a sketched hole trace as the input can now be placed on a curved plate.

TR281388
Duct flange dimensions are now considered for calculating a hole trace.

TR282244, TR282600, TR282601
New hole cuts now appear in the Part Monitor even if the manufacturing user does not have permissions to the hole trace/cut. (**Also implemented in 10.00.73.0077, 10.01.15.0081**)

TR282413, TR282742, TR282743
Holes existing on a seam now display correctly on manufactured plate parts. (**Also implemented in 10.00.73.0081, 10.01.15.0081**)

TR286153
All catalog hole fitting name rules are now supported and are working properly.

TR287485
If a user places a component on a pipe using a type from <Specify Component Tag>, a hole trace on this pipe will now be created.

TR290928
For a Multi-Cable Transit penetrating through Slab or wall hole trace is now created.

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HVAC

TR228480
Duct runs no longer distort when transitions are inserted between different cross-sections aligned to same axis, when one of them is routed in East, then North, then Elevation and then East directions.

TR263763, TR267834, TR267943
Selecting duct thicknesses having decimal values no longer have the thickness rounded-off to the nearest whole number. (**Also implemented in 10.00.73.0069, 10.01.15.0070**).

TR290053
User is now able to place ducts when duplicate permissible duct thickness entries exist in the catalog per material.

TR293737
The “Automatic Bolt Selection at Spec Break Rule” is now honored at thru-bolted connections.

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Interference Checking

TR165851
Notes on Interference markers are now retained if the interfering objects are moved to a new location where the clashes are still present.

TR171877
Invalid Interference markers no longer persist in the model after the reference file has been detached.

TR231175
Interference is no longer falsely indicated for two overlapping planes that are parallel (i.e. two pieces of equipment that are in surface contact with one another).

TR266906, TR266907
Interference Detection now allows the deletion of IFC entities in relation to invalid clashes with corrupted objects. **(Also implemented in 10.00.73.0066).**

TR270021, TR271874
Smart 3D now detects clashes between Smart 3D objects and Reference 3D model objects. **(Also implemented in 10.01.15.0072)**

TR270713
Interference Detection has been fixed to detect and report clashes for Maintenance-to-Maintenance aspects.

TR296127
.Net CustomIFC rule is not invoked by CodelessIFCRule when using 64-Bit IFC service.

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Materials Handling

TR270680, TR271026, TR271027
Modules and trusses can now be copied to any custom node under the Modules root in the catalog, and then placed using the Place Belt Components command. **(Also implemented in 10.00.73.0070, 10.01.15.0071)**

TR295920, TR296961
Now we can create curved trusses of radius greater than 1000m and up to 15km using Truss Wizard. **(Also implemented 10.01.15.0091)**

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Model Data Reuse / Model Data Transform

TR275914
Distribution databases, whenever they exist, are no longer displayed as selectable sites in the Model Data Reuse Wizard, as these databases are not eligible for these operations.

TR279727
Database incompatibility error messages will no longer be displayed when selecting models for Model Data Reuse operations across sites that exist in the same or different servers. Both cases are supported.

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Molded Forms

TR160068
Plates that are bounded by other plates with varying leaf thicknesses are now able to be bounded correctly.

TR165951
If the user selects Profile by Intersection, then selects the planar plate to create a profile on, and then for the second step user mistakenly selects the hull, the user will not receive an error and the command will proceed.

TR195209
On creating multiple plates by offset where modifying the offset value of one plate does not impact the offset value of other plates, the modified offset values does not get applied to all the plates.

TR203123, TR258894, TR258895
Sketched boundaries defined normal to the non-orthogonal plane with 45 degrees in the extrusion curve can now place in the expected location by Copy Symmetry/Mirror commands. **(Also implemented in 10.00.73.0058, 10.01.15.0062)**

TR207926
When a user unbounds a detailed profile, the profile is being updated to show the change to the end cut.

TR208273
If Profile 1 is constrained to Profile 2 by a connection point, and Profile 2 is moved, Profile 1 is now updated.

TR215062
In the model, a linear extruded plate no longer appears in the To Do List with the error message “error in bounding the plate” after using the Copy Symmetry command.

TR219236
When modifying the boundaries of a linear extruded plate (LEP) in a certain workflow, using the accept button, no longer causes all boundaries for the LEP to be lost. No To Do List message or other errors are generated.

TR219755
The logical connection between the plate edge and bracket is now created successfully if one of the bracket supports is a sketched plate boundary.

TR225534
When a user tries to create a seam by offset using the vertical method, in a case where the projection of the selected base curve does not intersect the plate surface, seam creation is now successful.

TR237680
When a bracket is created on a knuckled profile as a support, and if the knuckle is very close to second support, the bracket is practically connected to two leaf profile systems. There is now multiple leaf logical connections created, which means the assembly and physical connection is complete.

TR241499
The user selects a plate edge as the bounding object to both ends of a profile and now gets correct results depending on how the sketch was created.

- Case 1 – Continuous Sketch
  Three leaf logical connections are found to be created and no To Do List messages are found when the profile is detailed.

- Case 2 – Intersecting Sketches
  Multiple leaf logical connections are found to be created and results in just the profile end detailing correctly. There are no To Do List messages for this case.

TR242261
When a bracket is placed with an edge reinforcement as one of the supports and the edge reinforcement is split, the bracket shape or buckling stiffener do not change. The behavior is dependent on the distance of the seam point along the edge reinforcement from the other support.

TR243955
When an edge reinforcement on a plate (Plate-Edge Reinforcement combo as stiffener) is used as Support 1 and a stiffener as Support 2, the bracket shape is now correct.

TR246678
For a particular linear extruded plate in model, openings are now being deleted properly. When the opening is deleted, the detailed part updates properly, but the leaf and root system now reflect the deleted opening.

TR249320
1S bracket dimensions are now correct when the support is a profile cross section. In the bracket symbol parameter, all noses are 25mm.

TR249703
When a user tries to copy an opening from port to starboard side using the Copy Symmetry command, the opening is now copied.

TR251997
When Molded Forms support tools are loaded and accessed to utilize the To-Do-List Analyzer, an error is no longer seen and populates with To-Do-List items.

TR254367
The linear extruded plate, Type Deck, can now be bound to the knuckle profile. The profile is split at the knuckle. On giving the knuckle profile as boundary, the ambiguity for plate is now solved.

TR256175, TR263110
When using split root system (SRS) on a profile system with a defined orientation rule assignment along with defined first and second angle definitions, after split, the resulting two profile root systems will have now have correct first and second angle values. **(Also implemented in 10.01.15.0065)**

TR257080
When a profile present on a slanted deck is split and one of the leaf profiles is given as support to create a bracket, the shape of bracket is now correct. The shape of bracket is considering adjacent leaf profiles.

TR257306, TR259801, TR259822
When inverting the logical connection between a transverse bulkhead and the profile passing through it, the leaf logical connection between the two is no longer getting deleted. **(Also implemented in 10.00.73.0060 and 10.01.15.0064)**

TR257475, TR259246
When a profile is bound to a knuckled linear extruded plate such that the landing curve of the profile exactly coincides with one of the knuckled surfaces (coincides with the mother curve of the linear extruded plate), then the logical connection is now created. **(Also implemented in 09.01.30.0095)**

TR257730, TR258972, TR258996
Users can now multi-edit bracket properties on a workstation without programming resources. **(Also implemented in 10.00.73.0059, 10.01.15.0062)**

TR257755
If a bracket has as its supports tripping stiffeners, the toggle direction now works properly, and the direction of the bracket can be changed.

TR258469, TR275539, TR276281
A plate part is now trimmed when the thickness direction of the plate system is set to starboard. The trim is now perfect, so assembly and physical connections are now generated. **(Also implemented in 10.00.73.0076, 10.01.15.0077)**

TR260299, TR261979, TR261980
When a stiffener on a split knuckled linear extruded plate is used as support, the 2S bracket is now created as expected. Symbol debug shows that re-symbolization of the stiffener on linear extruded plate is now correct. **(Also implemented in 10.00.73.0075, 10.01.15.0064)**

TR260698
A profile is placed on a split hull and then is split by a natural knuckle point. When the profile is finished splitting, the root logical connection is no longer on the To Do List.

TR261459, TR261858
When updating out-of-date objects from Tools > Delay Settings, Update is now working, given the order is set to 'Optimal' in Delay Settings. **(Also implemented in 10.01.15.0064)**

TR261944, TR266159, TR266160
A bracket's nose length parameter value is now set by the short side in hull case. **(Also implemented in 10.00.73.0065, 10.01.15.0067)**

TR263204, TR264262
When creating a bracket, if the user changes the thickness direction from the default value, the thickness direction will now be the desired selection upon finishing the command. **(Also implemented in 10.01.15.0065)**

TR263252, TR264768
When placing a 3S offset bracket (BPS_3SL_OT), where one of the heights is measured as an offset from an edge reinforcement, the offset is now being measured from the edge reinforcement part. **(Also implemented in 10.01.15.0065)**

TR263462, TR266642
The name for a buckling stiffener system for 3S bracket is not missing when the third support is at a distance <800mm from the second. The stiffener is created and there is no longer a To Do List message stating "Failed to create/modify reinforcement." **(Also implemented in 10.01.15.0066)**

TR263539, TR264767
While trying to create a 4S sketched bracket using two plates and two profiles as support (plane by selected supports), the user is now able to select the 4th support. This is only for 4S sketched brackets, as 4S symbol brackets have never supported 'Plane by Selected Supports'. **(Also implemented in 10.01.15.0065)**

TR264805
When creating a new ParamText Geometric Construction, the error message box now only shows up after the user clicks finish and if applicable.

TR265117
When trying to place a tripping stiffener with the first boundary as an adjacent stiffener and the second boundary as free edge on the plate, a leaf logical connection between the tripping stiffener and free edge is generated. A horizontal seam should be placed on the plate on where the tripping stiffener is placed. As a result of generation of logical connection, the free end cut is now created.

TR266208
When creating a custom advance plate system macro using the 'GenericPlateDefIMP' template, the "category" parameter will no longer show up unnecessarily on the geometric construction ribbon bar. The "category" parameter only appears when there are multiple operations to manage.

TR266863
If a bracket is created using the Plane by 3 Points command, and if the supports are a plate and an edge reinforcement on one of the plate edges, the bracket can now be toggled.

TR270065
When a bracket is created with a hull plate as one of the supports and if the hull plate has a thickness offset, the bracket parameters are now adjusted to account for thickness offset.

TR272319
Users are now able to select an object graphically when a command shortcut key is used while actively in another command.

TR272469
Brackets are now created properly between the edge of the edge reinforcement and stiffener. There will not be a gap/penetration created between the bracket edge and edge reinforcement edge.

TR274832, TR282827, TR282829
While placing a tripping stiffener with the connection type connected at the starting point and the connection type Z offset at the ending point, the tripping stiffener is now placed correctly. **(Also implemented in 10.00.73.0081, 10.01.15.0081)**

TR280513
When the thickness of the leaf plates is changed and then the profile orientation of the Built-up can is changed in the parent plate property page, the thickness of the leaf can plates are no longer changed to their previous state.

TR282321
The name of a plate system will change when the plate geometry type is changed.

TR282770, TR283429
The advanced opening template, AdvancedOpeningImp.cls, required to build custom openings is now available. **(Also implemented in 10.01.15.0082)**

TR284988
When a user performs the Split Root System command on a profile using a plate as the splitter, no extra logical connections are getting created.

TR286867
Plates bounded to LEP and NLEP (created using GCs) are now able to trim correctly during detailing. **(Also implemented in 10.01.15.0087)**

TR287610
Ambiguity will now be correctly solved when user places profile on the hull where the boundaries are the hull on the port and starboard side.

TR288289
When user places a tripping stiffener with a perpendicular constraint on the anti-molded side of plates, the tripping stiffener will now be correct.

TR291157
When user bounds a Geometric Construction (GC) seam with another intersection GC seam, the seam will now place with no errors being generated.

TR291634
Brackets which previously failed to place at specific hull locations will now place correctly.

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Piping

TR174325, TR282506, TR282507, TR282508
The dimensional changes (width and thickness) for already placed reinforcing pads are now correctly reflected in the properties page after performing a synchronize model with catalog. (**Also implemented in 09.01.30.0120, 10.00.73.0084, 10.01.15.0084**)

TR175961, TR271083, TR274072, TR274073
Modifying the names of pipe runs by selecting multiple sloped and non-sloped runs, no longer results in the modification of the sloped direction to 'MULTIPLE'. (**Also implemented in 09.01.30.0110, 10.00.73.0074, 10.01.15.0073**)

TR195366
Bend radius multiplier is now getting updated appropriately when changing an elbow to a miter.

TR199588
Reportable parts are now displayed correctly in the properties page for mating parts.

TR225082, TR266658
The SmartSketch Nozzle option no longer needs to be turned off to move a pipe end feature to align with another pipe end feature. (**Also implemented in 10.01.15.0068**)

TR232270
The correct multisized component is now placed even when two multisized components have the same shortcode, optioncode, firstsizefrom and firstsizeto.

TR234846
To Do List entries for "Mismatch in port location" are no longer generated when placing a flange and then a pressure relief valve on the end of a vertically routed pipe feature.

TR236992, TR260053, TR260054, TR260055
Changing a pipe feature from insulated to not insulate now resets the thickness to zero. (**Also implemented in 09.01.30.0097, 10.00.73.0060, 10.01.15.0064**)

TR239023
The reportable part length is now getting updated for reinforcing pads after running the Synchronize Model with Catalog command.

TR239417
The Schedule Override field is now populated when beginning a new piping run.

TR251077, TR260283, TR260284
The user can now route vertically from a vertical pipe using a cardinal point offset. (**Also implemented in 10.00.73.0060, 10.01.15.0064**)

TR256025, TR258998, TR258999, TR259001
In a branching case, the header specification will always be used to generate the branching part. (**Also implemented in 09.01.30.0096, 10.00.73.0059, 10.01.15.0062**)

TR257943
The message displayed when trying to delete a piping parts now says, "Parts cannot be deleted. Select and delete the feature that generated the part."

TR258215
The extracted 3D Model Data for spools now reports the correct spool centerline length.

TR258499, TR260999, TR261000, TR261001
Routing from the starting point of a sloped pipe will now route the new pipe with the same slope direction. (**Also implemented in 09.01.30.0097, 10.00.73.0061, 10.01.15.0064**)

TR260207
Issues with deleting or modifying pipe runs due to false status and permissions errors have been fixed.

TR263687
Users can create a Reference 3D pipe run name label on a Reference 3D pipe.

TR269646
Insulation graphics now appear on 90-degree piping elbows created using the spec A10 downloaded from SmartPlant Reference Data.

TR270013
Users can now delete a clamp from the catalog and then bulkload the same clamp back without any issues.

TR273431
Users can now use the same session file for modeling when switching between models in the same site that have different catalogs.

TR275528
The gap between two along leg features is now maintained when copying and pasting just the features.

TR276092, TR278285
Flanges are no longer overlapping at run breaks. (**Also implemented in 10.01.15.0079**)

TR283240, TR287935
A vertical sloped run no longer changes angle when insulation is applied to the tap branch.

TR283355, TR284303, TR284304
Users can now connect two correlate pipe runs that form a closed loop. **(Also implemented in 10.00.73.0085, 10.01.15.0084)**

TR292419, TR293580
Flange thickness dimension for existing pipe stock now changes after changing the thickness on the bolted generic data sheet, bulkloading the changes, and then running Synchronize Model with Catalog. **(Also implemented in 10.01.15.0091)**

TR294619, TR296111
User is now able to place piping splits at the satellite location in a specific setup. **(Also implemented in 10.01.15.0091)**

TR298974
When a Default and an Optional pipe feature with different reporting types (in PCMCD) are routed, modifying the straight option from the ribbon bar now changes the 'Reporting Requirement' of the pipe part accordingly.

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Piping Isometric Drawings

TR100255
New Symbol keys have been provided (namely 4CYL, 4BOX, 4CYL3, 4BOX3, 4BOX4 and 4CYL4) for the delivered generic instruments.

TR158527
The character height in the Drawing Frame and Material List is no longer affected by the changes in the Pipeline Scale or Drawing Area Scale option.

TR196280
The label values (such as Spec Break and Insulation Thickness) are now fitting into the Text Box border.

TR201894
The sheet number format is now respected on the Continuation label by showing a leading zero along with the sheet number.

TR202388
The duplicate Component Note conditional labels are not displayed on the isometric drawing if the labels are mapped under the same LabelAttribute2 Type.

TR218326
The isometric drawing is now successful when the field liner thickness and gasket thickness are defined for the pipeline.

TR227219
The Correct Continuation coordinates are shown at the tee branch and header when the LengthtoBranchPoint option is set to True.

TR229253
Supports are now shown on the bend point locations rather than at the center point of the bend in the isometric drawing.

TR232810
The Support label now points to the center location of the instrument and valve with 01HG SKey Mapping.

TR233123
The embedded labels for nuts and washers are now shown on the isometric drawing

TR236451, TR244840, TR252048
Isometric drawings now report the support in the material list when a combined support is placed on two different pipelines. **(Also implemented in 09.01.30.0090, 10.00.73.0053)**

TR246366
The Detail Sketch redundant option values .dwg, .dxf and .cel are now removed.

TR252437
The implied bolts and gaskets are now shown in isometric drawings when a reducing flange is placed at an equipment nozzle and piping is routed in different planes.

TR252747
Modifying the miter weld Atext -457 now brings the correct text and miter angle on isometric drawings.

TR253966, TR265593, TR269690, TR273067
Isometric drawings now show the correct Insulation break values at the reinforcing branch weld portion routed from the header line. **(Also implemented in 09.01.30.0108, 10.00.73.0071, 10.01.15.0062)**

TR255393
The Jacketed Isometric is now successful when a Half Coupling component is inserted on the Jacket Pipe.

TR257547
Unwanted question marks no more appear when -ATEXT 318 option is enabled with text value more than 13 characters.

TR258945
The correct weld drawing number is reported in the weld list file when the material list overflows to a new sheet.

TR260430
Isometric drawings are successful when the Field Lining Pipeline is containing through bolted components.

TR263111, TR264721, TR264722, TR264729, TR264730
Isometric drawings no longer show the solid continuation line when a trunnion support including taps are placed on pipe parts. **(Also implemented in 09.00.71.0080, 09.01.30.0099, 10.00.73.0064, 10.01.15.0066)**

TR263377
The break point using control points at elbow portions are coming out correctly and the isometric drawing does not show half of the elbow on two sheets of the drawing.

TR263532
'USER DEFINED FITTING SPLINDLE KEY ???? NOT FOUND' error no longer appear in .MES file for each and every isometric even though the update is successful.

TR267650
The last drawing frame modifications done in the Options browser through the drawing set up tool are now getting saved to the catalog.

TR268483
The delivered variable layout weld list box is now properly formatted without any overlaps between column headings.

TR268688, TR269686, TR269687, TR269688, TR269689
The start coordinates are now written to the PCF when pipes are broken by field welds at module boundary with Change Management enabled. **(Also implemented in 09.01.00.0100, 09.01.30.0104, 10.00.73.0068, 10.01.15.0069)**

TR268840
Isometric drawings are successful in splitting the isometric at the right point when a control point is placed at the distributed connection of the tap branch of an orifice flange.

TR272936
The Isometric now shows the user-defined Material description for clamps.

TR273168
The Isometric is now able to generate a 5-port instrument using the redefined Symbol key IR**.

TR274419
The Pipeline Cross Over section is now represented as a 3D Vector Hidden Line drawing.

TR274798
Skew dimensions are now shown correctly when a pipe is connected to a bevel end nozzle and Column Reference is enabled.

TR275096
Unwanted question mark symbols are no longer reported on isometric drawings on a valve spindle direction message.

TR275099
Short stub lines are not created when a pipeline is routed with components having socket weld connections.

TR276161
No more missing data for Supports in the PCF file even if the drawing class option is set to “System” for WBS Isometrics.

TR279422
Now the material user attributes are listed in the PCF even when the active material list is fixed.

TR280077
Flange, Bolts & Gasket callout no longer appears separately on Field welds which has the support placed on that Straight feature.

TR282919
The Socketed welded symbol is now coming out correctly on the by-pass line of Pipeline Isometric drawing.

TR283777
The Support label is now pointing to the center of the reducer instead of the support connect point.

TR284638
Isometric drawings no longer fail to update when the option ‘S3D.Drawing.Content.TreatTapsOnPipeAsTeeStubs” set to TRUE when there are more number of TAPS on the pipeline (for example: more than 60).

TR284772
Isometric drawings are now successful if a pipeline contains a T strainer and a split point is placed on the port of the T strainer.

TR285587
No more incorrect dimensions when ‘Gaskets’ are set to ‘Included’ option.

TR286187
The Jacketed continuation section is now shown in dotted lines.

TR286837
Isometric generation for jacketed pipelines no longer fails when ATEXT 557 and 558 are blank.

TR287228
Isometric no longer fails for a very long pipeline which has more than 30 sheets.

TR288754
WBS Isometrics for Jacketed Piping are extracted correctly without unnecessary data such as core piping.

TR290401
Slope value appears correctly even for small slope values such as 1:10000.

TR299341
New option “Include Bolts with No Gaskets” has been added which allows the bolts to be reported in S3D neutral file even when the gaskets are undefined.

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Planning

TR155846, TR276191
After splitting a planning joint, the original planning joint does not appear in the GUI or workspace explorer.

TR169506
When splitting a block, cutting planes are now visible after resuming the command and entering the Select Cutting Planes step.

TR244033
Block assignment now allows consecutive runs without restarting the command.

TR244306
When planning joints are moved to higher assemblies to postpone the welds, a type mismatch error no longer appears. The planning joints are moved to the assembly selected.

TR245039
Planning joints now have new logic that take into consideration the weld orientation and rotation for calculation.

TR245440
User created a new Assembly under Root Project, changed the Assembly Type to Assembly Block. With Assembly Block under root project user selects property page, the property page now opens.

TR247712
Show Assembly method now works correctly; it displays Stage, Type, Weight, Length, Width and Height.

TR259309
Users can now assign an assembly as a child to another assembly.

TR260213, TR269707, TR270028
When users change the Assembly Type to Modular Assembly on the properties page, the task no longer becomes corrupt. (**Also implemented on 10.00.73.0068, 10.01.15.0069**)

TR260643, TR260807
In the Manufacturing Service Manager, production routing changes are now reflected on the part as “Out of Date”. (**Also implemented on 10.00.73.0060**)

TR261616, TR262004, TR262005
Property pages are now enabled in the Create Assembly command. (**Also implemented on 10.00.73.0062, 10.01.15.0064**)

TR268071, TR268280, TR270106
Show Assembly Method/View Orientation now displays correct orientation in the graphic view. (**Also implemented on 10.00.73.0068, 10.01.15.0070**)

TR266799, TR270509, TR270510, TR270511
When new parts are added to a common part group for the second time, the group is no longer getting deleted upon refresh. (**Also implemented on 09.01.30.0105, 10.00.73.0069, 10.01.15.0070**)

TR268868
Common Part Service performance has now been improved.

TR272391, TR275265, TR277008
Show Assembly Method now has no errors after sequencing, regardless of which level in the assembly is selected. (**Also implemented on 10.00.73.0077, 10.01.15.0078**)

TR274660, TR278137, TR278346
UpdateWCG function no longer requires the user to have permissions on a HgrSupport or HgrDesignSupport. UpdateWCG also now works properly if the Status is set to 'Approved'. (**Also implemented on 10.00.73.0077, 10.01.15.0078**)

TR274266, TR275235, TR275236, TR275237
Common part definitions are now retained when the user re-computes the assembly. (**Also implemented on 09.01.30.0112, 10.00.73.0075, 10.01.15.0076**)

TR277679
Show/Hide feature now functions properly for Reference 3D objects in the Planning environment.

TR279473
Equipment components are now assigned to the appropriate block after running Block Assignment.

TR281556
Global and assembly orientation range values are now correct for Assembly in model.

TR281579, TR282284, TR282285
After re-running Common Part Service, existing group names are no longer being removed in existing groups. Creation dates are also no longer being modified. (**Also implemented on 10.00.73.0080, 10.01.15.0082**)

TR281923, TR283312, TR283313, TR283314
Sessions no longer crash when custom processing plate parts. (**Also implemented on 09.01.30.0120, 10.00.73.0083, 10.01.15.0082**).

TR283126, TR283494, TR283496
When placing an orientation rotation on an assembly and selecting the Show Assembly View command, the displayed view is now correct. (**Also implemented on 10.00.73.0082, 10.01.15.0082**)

TR283068, TR284056, TR284057
Common parts are no longer lost when a part is moved into an assembly. Now parts are under respective common groups after moving parts from an unassigned folder to another assembly once Common Part Service is run. (**Also implemented on 10.00.73.0083, 10.01.15.0083**)

TR284388
Performance has been improved for Assembly Range computation.

TR289044
Common parts are now listed in the common part group and the remaining parts in the assembly are under a unique folder. (**Also implemented on 10.00.73.0079 and 10.01.15.0060**)

TR295796
User now able to generate Planning report which has calculates the Surface area in model. For example, the report 'Assembly and Parts With Surface Area No COG' and 'Block Assembly and Parts Report With Surface Area and No COG' will display Surface area.

TR296283
In model with Tee Welds of 300 mm at block intersections, selecting Generate Planning Joints, picking any assembly and Finish will now not crash the session.

TR301237
When user executes Tools > Run Report > Types of Reports > Planning > Common Parts Report which provides applicable data from the Common Parts Service Results, this data is now derived from the applicable Part Object property values.

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Project Management

TR193180
PDMS equipment is positioned as expected when attached as a Reference 3D model.

TR217819
Database conversion errors from Oracle to SQL are resolved now.

TR220354
DGN files are shown up as expected when attached as a Reference 3D model.

TR226860
If the Interference Server node on a model is missing when creating the model, a warning message appears stating that the Interference Server node failed to create. There is a custom command available to create the Interference Server node when it is missing.

TR234864
Referenced plate and profile parts have correct data in Reference 3D properties.

TR238960
Referencing a ZVF along with its mdb2 as a Reference 3D model using SPIOP filter displays no errors.

TR249293
Satellite location shows Reference 3D model as up-to-date after re-attaching at Host

TR253872
The synchronization with SQL Management Studio has been enhanced to account for time zone and daylight savings time changes.

TR256367
MicroStation v8 DGN files are now showing up correctly when referenced as Reference 3D models.

TR257788
Reference 3D models translated by SmartPlant Interop Publisher from a PDS sample dataset now successfully attach to a Smart 3D model.

TR259803
Ship property numeric values displayed in Project Management will reflect the proper units for those values.

TR263275, TR263542, TR263543, TR263544
In a Global Workshare configuration (GWC), Database Integrity can now only be run at the host location. Any Database Integrity records from the host are then replicated to the satellite locations. This is done to prevent objects from losing their relationships to the system hierarchy or parent folders in the workspace. **(Also implemented in 09.01.30.0101, 10.00.73.0063, 10.01.15.0067)**

TR263688, TR264980, TR264981
The setup of two or more Oracle GoldenGate models on the same server for Global Workshare purposes has been enabled. This fix applies to setups of multiple models within a site or in different sites. **(Also implemented in 10.00.73.0064, 10.01.15.0066).**

TR265839
The command to upgrade databases has been fixed so that one object failing to upgrade will not cause several other objects to fail to upgrade, when those objects have no issues preventing them from successfully upgrading.

TR270612
Correct graphics are now shown for certain DGN files attached as Reference 3D models.

TR273092
ZVF files update properly using Reference 3D update and schedule functionality.

TR273473, TR274060, TR274061, TR274062
The Database Integrity command was cleaned to avoid a rare corruption problem that used to be caused by addressing duplicate flavors. **(Also implemented in 09.01.30.0110, 10.00.73.0074, 10.01.15.0073).**

TR274561
Reference 3D projects submitted from a client are getting updated on the batch server as designed.

TR275801
If the user changes the Windows regional settings from a non-English language to an English language after attaching a Reference 3D model, the measurement settings for that Reference 3D model (i.e. reference point location) will not be rounded off, resulting in an improper value.

TR280670, TR282503
If the user runs the Check Data Consistency (also known as Database Integrity) custom command (SP3DCheckDatabaseIntegrity.CCheckObj) in the Smart 3D workspace and then runs the Clean Database custom command in the Smart 3D workspace (SP3DCleanDatabaseCmd.CCheckObj), the user will be able to view site, catalog, and model databases in the Clean Database window, rather than only the site. **(Also implemented in 10.01.15.0081)**

TR287059
Reference 3D mapping no longer fails when the source object part class name is listed in the .drv file from PDMS before the "Type" attribute is encountered.

TR289989
Users with read-only or no permissions to the catalog are no longer able to delete items in the catalog.

TR291056
Batch queues are now displayed in a Workshare environment.

TR30071, TR298676
Converted IFC files through SPIOP are now able to load and update Reference 3D data. **(Also implemented in 10.01.15.0085)**

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Reports

TR159583
Users can now configure 'Macros' on multiple sheets of a report.

TR160082
Users can now write an SQL query that returns the Name Rule used by an object on the reports database.

TR161338
Successive Recursive Strokes in a COM label query now works. After completing the first recursive stroke navigation, the software uses the end result of the first recursive stroke before navigating further.

TR162133
An additional '\' character will no longer be added when a recursive label with concatenate '\' is used to create the drawing/system hierarchy labels. Earlier it used to give '\\SP3DTrain\\Drawings\\Composed Drawings'. Now it gives without the extra '\' as \SP3DTrain\Drawings\Composed Drawings.

TR197417
Structural Bill of Materials (BOM) reports are now compatible with the Smart 3D report template editor.

TR233304, TR259421, TR259753, TR259825
Reports framework now supports .xlsm format as well as .xls and .xlsx. **(Also implemented in 09.01.30.0096, 10.00.73.0059, 10.01.15.0063)**

TR254937
Corrugated bulkheads' length and width parameters are now being reported in reports.

TR265603, TR272839, TR272840, TR272841
Piping Isometric Component Part Spec Note label now reports all the notes that were defined on the Part definition. **(Also implemented in 09.01.30.0110, 10.00.73.0076, 10.01.15.0075)**

TR267029
Modifying a label and testing it as tool tip no longer requires reopening the session file.

TR272837, TR272093
Delivered SQL query Labels will now get resolved in Oracle databases such that users can use them in Reports, Isometrics etc. **(Also implemented in 10.01.15.0073)**

TR286708
Property-based queries are now working appropriately in reports when an object no longer exists in a drawing view.

TR287132
If a report fails to update and requires regenerating the reports database, the user will now see the following message in the error log: "Failed to execute the query on report database. Please try regenerating report Database again to reflect the changes done in catalog or model database."

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SmartPlant Enterprise® Integration

TR211962
Verify PID custom command can recognize drawing items on the PID now.

TR212789
Issue of upgrading schema modifying SupportedSchemaVersions in SP3DPublishMap.xml is fixed.

TR221364
Delivered Tool Schema file now has correct tool version information.

TR227211
Design Basis time stamp is now updated before clicking OK when receiving message box indicating that the document is retrieved successfully.

TR257074, TR259483, TR259484, TR259485, TR259486
The Find Documents to Publish command no longer takes a really long time to display the dialog when there are a large number of documents published. **(Also implemented in 09.00.71.0103 with SmartPlant Client 4.4.4.0001, 09.01.30.0097, 10.00.73.0060, 10.01.15.0063)**

TR257078, TR270958, TR270959
The view cable schedule is no longer showing the wrong list of cables. **(Also implemented in 10.00.73.0070, 10.01.15.0071)**

TR261017, TR263208, TR263209, TR263210
3D Model Data XML generation in Russian Homogeneous environment with '.' as delimiter is working now. **(Also implemented in 09.01.30.0099, 10.00.73.0063, 10.01.15.0065)**

TR261822, TR262775, TR262824
Smart 3D is able to publish a view file for drawings with embedded report. **(Also implemented in 10.00.73.0063, 10.01.15.0065)**

TR262544
VolumetricFlow UOM is now added to Design Basis Schema.

TR265879
There are no longer issues of missing entries in the delivered SPRDirectToR3D Mapping XLS file Target Interface and Target Property columns.

TR266962, TR267858, TR267859, TR267860
Published files no longer have FileToolData XML file of only 1KB when publishing interactively. **(Also implemented in 09.01.30.0103, 10.00.73.0068, 10.01.15.0070)**

TR268084
An ampersand character in the 3D Model Data document name no longer causes insulation not to appear in VUE file.

TR274810
User is able to revise group documents to the next major revision.

TR275360, TR275834, TR275835
After disabling anonymous access and enabling Windows authentication on SmartPlant Foundation (SPF) server, Smart 3D now is able to communicate to the SPF server. **(Also implemented in 10.00.73.0075, 10.01.15.0076)**

TR275657
The issue of generating design basis getting Duplicate Column Name error is fixed if there are attributes in the component schema file having same name at a length which is more than 30 characters.

TR276413, TR279494, TR269495
Smart 3D no longer renames assemblies with their parent's assembly. **(Also implemented in 10.00.73.0081, 10.01.15.0081)**

TR276737
Type Mismatch error no longer displays when upgrading schema.

TR292294, TR292733
Publish log file is enhanced to provide more summary information. **(Also implemented in 10.01.15.0090)**

TR294980, TR294527
Schema Component has capability to validate Shared Content UNC path. **(Also implemented in 10.01.15.0091)**

TR298437, TR292924
Update Compare Design Basis is able to reflect cable connected equipment now. **(Also implemented in 10.00.73.0093)**

TR301499
Correct external UID of SPEL cable is able to be published.

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Space Management

TR234697
Egress Volume disappears when cardinal point is changed or path is modified

TR238296
To-Do List entries created for space volumes for missing relationships with catalog parts are cleared upon update.

TR286513
Unable to clear To Do List entry 'No catalog part is associated to Space Entity'

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Structural Detailing

TR150683
Any part can now be used as reference in Sketch2D for creation of lapped plate.

TR169130
Plate and profile edge reinforcements are being supported by copy/paste.

TR205967, TR283831, TR284567
Brackets are created successfully when one support is a BUL_3 and the thickness of the bracket is equal to or greater than the thickness of the web. **(Also implemented in 10.00.73.0084, 10.01.15.0083)**

TR212598
The split point location is now correct when splitting a physical connection using Offset Method Girth with the reference plane at same location as profile.

TR222449, TR268052, TR268054
Profiles with planar angles placed on bent plates with small angles (177-180 degrees) the profile will now detail. **(Also implemented in 10.00.73.0066, 10.01.15.0068)**

TR224640
When a physical connection is split by a straking seam, the leaf physical connections are created between plate parts of different thicknesses.

TR231864, 261887, 261888
A Geometric Construction knuckled plate with an opening now trims successfully. **(Also implemented in 10.00.73.0062, 10.01.15.0064)**

TR232153
Trim no longer fails for a deck bounded to a bulkhead's molded surface, offset from edge boundary.

TR243959, TR259738, TR259739, TR259740
Copy/paste maintains modified values of plate detailing features. **(Also implemented in 09.01.30.0097, 10.00.73.0060, 10.01.15.0064)**

TR247066
Users can split physical connections correctly at 90 degree corners.

TR247209, TR261329, TR262890
Knuckled Flat Bar now details when the landing curve is on the bend side. **(Also implemented in 10.00.73.0063, 10.01.15.0064)**

TR254273, TR265852, TR265853, TR265854
Assembly Connections are generated between a plate and profile when the Logical Connection is inverted. **(Also implemented in 09.01.30.0101, 10.00.73.0065, 10.01.15.0068)**

TR256017
Built-up Braces with AxialGap-Both frame connections detail successfully.

TR256395
Plates connected to spud cans (such that they cross knuckle) are trimmed correctly and result in the necessary connections.

TR257057, TR263449
Assembly connections are generated for profiles bounded to a hull with ambiguity solution and "Up-Down" orientation. **(Also implemented in 10.01.15.0065)**

TR260102, TR262101, TR262102
Users can split physical connections by offset method when chamfer and corner features are on the part. **(Also implemented in 10.00.73.0062, 10.01.15.0064)**

TR261799
User modified bevel values of existing physical connections are now reflected correctly on the ribbon bar.

TR262117, TR264333, TR264334
Users can place edge treatment on a hole cut on a part. **(Also implemented in 10.00.73.0063, 10.01.15.0065)**

TR262377
Assembly and physical connections are now created when a ruled plate is bounded to a linear extruded deck plate.

TR263114
Physical connections created are correct when the split point is close to an edge feature.

TR268347, TR272891, TR273091
End cut shapes are created correctly at the plate and profile knuckle junction. **(Also implemented in 10.00.73.0071, 10.01.15.0072)**

TR269579
Slots and collars are created when a plate is penetrated by a web plate and edge reinforcement.

TR266278, TR269880, TR272128, TR272129
Plates with slots at a corner detail correctly. **(Also implemented in 09.01.13.0104, 10.00.73.0071, 10.01.15.0071)**

TR266582
When creating an advance plate system macro using geometric constructions, inputs can now be hidden from the ribbon bar.

TR270872, TR271873, TR271906, TR271907
A plate system with an edge reinforcement that penetrates another plate system, now creates slots and collars when detailed. **(Also implemented in 09.01.30.0108, 10.00.73.0072, 10.01.15.0073)**

TR273759
Users can modify edge treatments placed on a knuckled plate with sketched boundaries.

TR274887, TR278574
A plate trims when bound to a deck and a standard member. **(Also implemented in 10.01.15.0078)**

TR280129, TR280276
When using the offset property in the Molded Conventions tab of a leaf plate, connected stiffeners will be taken into account if the offset is set to a negative value. When the leaf plate offset property is updated to a negative value, attached profile parts will get updated. This has fixed widespread problems with any slots and collars generated around the profile. **(Also implemented on 10.01.15.0079)**

TR280433, TR281311
Symbol sharing has been disallowed for high-risk detailing occurrences.
(Also implemented in 10.01.15.0080)

TR282683
When modifying the permission group for one leaf (detailed) profile, the other leaf part no longer receives the same permission group change.

TR288323
Profiles now trim when split at linear extruded plate knuckles.

TR289077
When detailing a transverse profile that has a longitudinal profile bounded to it, no To Do List messages are shown.

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Structural Manufacturing

TR195281
The plate length of a linear extruded plate is now correct in the XML file.

TR200477
Issues observed with the new SNU unfold algorithm on hull plates with seams are resolved.

TR238856
Various SNU unfolding issues are resolved with roll line, contour display, manufacturing tab, girth length, tangent lines, and neutral surface.

TR240718
In certain cases, when a chamfer is applied on the base port of the plate, during creation of margin, multiple edge ports are no longer available for placement of margin. Instead, there is a single lateral face on which the margin is applied (Multi lumps).

TR242277, TR252105
When unfolding a plate using 'SNU', 'SNU_Box', the roll line position is now correct.

TR244167
The delivered tool tip “Struct Mfg Tooltip MfgPlate” is now displaying the assembly name and block name. It is not showing “DATA INVALID” at these fields.

TR245501
In customer model, when unfolding a specific plate with ‘SNU’, tangent lines are now correct.

TR247677, TR248054, TR248056
Users are now able to manufacture a corrugated bulkhead with a rectangular opening. Corrugated bulkheads with sketched features also pass. **(Also implemented in 09.01.30.0090, 10.00.73.0053)**

TR248409
In SNU Unfolding: Oscillation issues with hull plates - wavy edges after manufacturing have been corrected.

TR248906
Roll Boundaries are now correct on revolved plates in part monitor.

TR252124, TR252850, TR252853
Profile bending lines are now correctly located when a fabrication margin is added to either end. **(Also implemented in 09.01.30.0092, 10.00.73.0056)**

TR252643
Tangent lines and marking lines are now present in part monitor output for plates.

TR253955
Through the part monitor output of a particular Built-up T hull profile, the Topflange_builtup_connection mark line now has a normal shape.

TR254400, TR254831
When a user tries to edit and rotate the Annotation in Annotation Editor, the function now works. **(Also implemented in 10.00.73.0055)**

TR254703
Plate unfold is now correct using SNU - PlateUnwrapAlogrithm and Detailing Neutral Surface - Surface Type Requested for Unfolding.

TR255270
The issues are resolved with manufacturing profile macros, with offset value, when margin is used.

TR255452, TR256536, TR256537
Manufacturing plate contours are now correct when the edge feature is near a hull tangent. **(Also implemented in 10.00.73.0057, 10.01.15.0061)**

TR256791
When a user unfolds the part using 'SNU' unwrapalgraithm, Plate Upside as Molded Side has correct outer contours.

TR257885, TR257976, TR257977, TR257978
Openings no longer cause the XML of swage bulkhead to be incorrect. **(Also implemented in 09.01.30.0094, 10.00.73.0058, 10.01.15.0061)**

TR257993
Part editor can now be opened if the user does not have write access on shared content. No error messages are shown.

TR258195
A manufactured plate part with fabrication margin applied does not error in structural manufacturing when selected with the workspace filter: All Manufacturing Entities.

TR257898, TR258817
When "Webleft/Webright for DSTV" is used as upside for manufacturing a member with a rectangular box (HSSR) cross section, then XML data is now generated. **(Also implemented in 10.00.73.0060, 10.01.15.0062)**

TR258401
During the initial assembly Pin Jig creation, the user defines boundaries. When the display list is activated and coordinate systems are selected from the Pin Jig boundary list, the selected coordinate system is now being shown in the graphics view.

TR258815, TR258819, TR258327
When a non-circular opening (rectangle, triangle, etc.) is placed on the web of a HSSR member, on manufacturing the member there are no longer issues observed with openings. **(Also implemented in 10.00.73.0058)**

TR258897
The curved plate part output is now correct in part monitor. The inner girth lengths are the same now when using detailed part surface and detailed neutral surface.

TR259197, TR262294
When a manufactured plate part is undetailed, the light plate part shows a '+' sign in the icon in the Workspace Explorer to denote that the plate part is manufactured. **(Also implemented in 10.01.15.0064)**

TR259640
Manufactured plate parts are now marked as out of date when the connected profile part name is changed in Manufacturing Service Manager.

TR260103
The curved plate unfolding length is calculated correctly when comparing girth length to surface length.

TR260622, TR264108
Users can now select the bracket either from the Workspace or from the Workspace Explorer to apply margins on bracket edges. **(Also implemented in 10.01.15.0065)**

TR261062
SNU unfolding for outer contour of plate has been corrected for oscillations.

TR263853, TR265326
Users are now able to select the bottom line of an LEP template to modify. The bottom lines are now highlighted in green. **(Also implemented in 10.01.15.0066)**

TR264521, TR266549, TR266550
A system is now able to create a penetration mark for a profile penetrating a plate when the connected tripping element is a stiffener. **(Also implemented in 10.00.73.0065, 10.01.15.0067)**

TR264645, TR265146
Detailing features on a profile are now shown in the part monitor output. **(Also implemented in 10.01.15.0066)**

TR264658
When user checks the manufacturing part monitor output for a part with 'Developable' algorithm, the results are now correct for the outer contour.

TR264862
Part monitor output for a standard member is now correct when the member is placed with an angle of 180 degrees, and on the property page option "Profile Upside" in the Process tab is set to "Webleft for DSTV".

TR265245
When the user checks the manufacturing part monitor output of a plate created with Geometric Construction, the knuckle marking line is shown in the manufacturing output.

TR265256
When a Tubular member is placed such that one of its ends is connected to an inclined plate, the user can now place the template at the end where the member is connected to the inclined plate.

TR265575
The correct shape outer contour is now seen in part monitor output using the 'Flat' algorithm, resulting in the correct manufacturing output.

TR265577, TR265618, TR265619, TR265620
When manufacturing a plate with Profile Marking Rule enabled and the profile cross section has special characters, there is now XML data. If the Profile Marking Rule is set to Ignore, manufacturing XML data is generated without the profile marking information. **(Also implemented in 09.01.30.0100, 10.00.73.0064, 10.01.15.0066)**

TR265682, TR280847, TR280848
After executing manufacturing on object (Plate) which has a width equal to 100 mm or less, the output data can now be shown in Part Monitor and Annotation Editor. **(Also implemented in 10.00.73.0079, 10.01.15.0080)**

TR266126, TR266388, TR266390
When a user creates an edge treatment from the Place Edge Treatment command in the Structural Detailing task on a plate with the hole cut in Hole Management, the manufacturing plate part now shows bevel information. **(Also implemented in 10.00.73.0065, 10.01.15.0067)**

TR266190, TR266353
When two templates are created on the same plate part, both the template marking lines corresponding to the created templates are visible in part monitor output. **(Also implemented in 10.01.15.0067)**

TR266379, TR268145, TR268146
It is now possible to have both annotations and symbols within the same part monitor view. **(Also implemented in 10.00.73.0066, 10.01.15.0068)**

TR267157
If any manufacturing parts get edited in Part Editor, the user can now perform modifications on that part and changes are reflected in Part Monitor on the manufacturing update.

TR267124, TR268320, TR268321, TR268322
For a T girder, the web plate bearing the flange output shows the edge bearing the flange is always on top, but this now passes when the "Web" height is more than the length. **(Also implemented in 09.01.30.0103, 10.00.73.0067, 10.01.15.0068)**

TR268282, TR269693, TR269694, TR269395
The unfolding length of the corrugated plate is now comparable to 3D. There is no longer a ~5mm difference in the length as observed between the 3D/2D and Part Monitor outputs. **(Also implemented in 09.01.30.0104, 10.00.73.0068, 10.01.15.0069)**

TR270324
In Part Monitor, the contour of a profile part is now closed. These results are now correct in Part Monitor & XML output for the manufactured profile.

TR270601
When a user unfolds the part using 'SNU', correct contours and marking lines are observed.

TR271260
When corner features are applied where two plate parts come together along a panel edge, several problems are resolved for the panel.

TR271382, TR273412, TR273413
There is no longer an issue of missing bevel details and marking at corner feature of cross section T, after the input of fabrication margin. **(Also implemented in 10.00.73.0073, 10.01.15.0073)**

TR271359, TR280665, TR280666
The Part Monitor output for H-Beam and I beam is now correct. **(Also implemented in 10.00.73.0079, 10.01.15.0080)**

TR272481
When a deck plate that is created having cornered edges is manufactured, the user now sees the correct outer contour in Part Monitor.

TR273082
When a plate with openings is manufactured and viewed in the Part Monitor output, the openings now have the same number of segments. The algorithm is producing equal segments for circular and for elliptical openings.

TR273246
For a Linear Extruded Plate created using a sketched boundary, the structural manufacturing XML data now being created.

TR273496
When a user creates templates for a hull plate and compares the template sketches with a hull plate part in the Molded Forms task, the user observes that the template sketches now match the hull plate part curve. The view vector is now computed correctly when the view direction is changed and hence no issue with the transformation matrix.

TR273530
After merging two marking lines, when a user tries to merge another line to it, an error message stating “Objects cannot be merged because they are not end connected” is displayed.

TR273920
Curved Hull plates in a model are no longer coming back as flat when manufacturing, and so the correct process and marking rules are used to process the plate output. This now results in correct manufacturing data.

TR274014, TR274757, TR274758
A user is modifying the annotation in Annotation Editor. After that, the user places a hole in a detailed plate part and in doing a Manufacturing Service Manager update for the plate, the annotation now shows modification. **(Also implemented in 10.00.73.0076, 10.01.15.0075)**

TR275006
When a member is created in such a way that a face has multiple disjoint segments, the DSTV output is now generated correctly.

TR275363
When unfolding a shell plate with SNU algorithm, correct outer contour of the plate is observed

TR275519
When plate has two features with the same name and fabrication margin added to the edge, no error is seen when user tries to open manufacturing property of a plate part.

TR276136, TR280667, TR280668
Structure is modelled such that the two profiles on the deck are passing through a Transverse bulkhead and are on the either side of the center line of the Ship (Port and Starboard side). The manufactured profile is taken into Annotation Editor to extend the plate marking on profile and the thickness direction of the plate marking on profile is changed on the correct side. **(Also implemented in 10.00.73.0079, 10.01.15.0080)**

TR276171
Regional margin is no longer ignoring the "region" and not applying to the entire plate's edge for a plate with curved ends.

TR278249
The accumulated shrinkage on a panel is now resulting in the correct panel length. Individual part shrinkage must be calculated so it can be applied at the panel level. By the time the panel is cut, a single shrinkage factor should be defined for the panel's primary and secondary directions.

TR278411
When a user tries to give the marking on the plate with a side as Anti-Molded, a glyph or arrow mark is shown in the Workspace. The direction is now correct and correct Part Monitor output of the marking side is shown. Marking is now shown in the dashed line when the side for the marking is selected as Anti-Molded side.

TR278564
When a user creates a box template with template marks as SlotAsCut, and checks the slot size in Part Monitor, the size of the slot no longer varies.

TR278910
Manufacturing Service Manager updates no longer generate errors caused by a pin jig whose work center was invalid (missing).

TR279564
Panel Line plate object in now not missing or has incomplete marking lines for attached profiles and plates.

TR279967, TR280662, TR280663
Users now have ability to assign a bevel with type Bevel-Custom in the Part editor so it is reflected in the XML output. **(Also implemented in 10.00.73.0081, 10.01.15.0081)**

TR280026
When a user tries to check the output of Manufacturing Template Object type from Tools-Custom processing command in Structural Manufacture task, all template reports are now reporting data in the output log files.

TR280132, TR282632
When member end cuts are being applied and the resulting member end cut feature type is applied to a member web that only cuts the fillet portion of a web, the feature is now being cut. **(Also implemented in 10.01.15.0081)**

TR280278
When multiple regional margins are applied on the same portion of the edge, the manufacturing output is now correct. One of the regional margins (randomly) is no longer applied on the whole edge port in the manufacturing plate output.

TR280325
If a manual chamfer is placed on an edge of a plate with no physical connections added (as there is no beveling needed), the bevel parameters on the edge are now correct in the manufacturing plate output.

TR280363
When a plate is created such that there are two curved portions in the mother curve without any flat portion, two sets of roll boundaries will be seen.

TR280711
When the user sets the pre-nesting process as Pre-nesting (form detailing definition, if applicable) and checks the part Monitor output, the width of the plate is now considering the long point length and giving the correct width when the part is detailed or un-detailed.

TR280874
The Panel Line XML output has multiple SMS_PLATE_CONTOUR node, but is no longer missing "process" in the XML file.

TR282254
User has short, bent profile that require “clamp” margin of ~1000 mm be applied to the ends. User is setting the type to “reference”, which means that it is now being processed when the manufacturing part geometry is created.

TR282992, TR283243, TR283245
A user can now perform an update/recompute/modify of a manufacturing part with current user permissions. It has been found that permission groups are no longer mismatching between manufacturing profile part and manufacturing geometry 2D objects. **(Also implemented in 10.00.73.0083, 10.01.15.0082)**

TR283222
Manufacturing the XML REFERENCE_SIDE output is now correct for flange (specifically for Bulb Flange).

TR283969
Panel Export adjusts the Panel-Margin for post welding cutting shape. The user is able to reflect the contour 'to-be-cut' as part of the final panel cutting process.

TR284334
Part monitor output is now correct for a conical shape plate part using Developable algorithm.

TR284439
Manufacturing xml is now generated when detailing neutral surface is used for unfolding.

TR284472, TR284840, TR284841
When a line-string (collection of lines/arcs) is exposed in the XML, Smart 3D no longer exports the definitions as “Arc.” **(Also implemented in 10.00.73.0084, 10.01.15.0084)**

TR284993
Managed export generates XMLs in the output folder if the space exists at the end of the Assembly name.

TR285055
Duplicated marking lines and duplicated annotations (at the same location) for base control marks and template marks no longer appear on a panel.

TR285730
In customer model, an inclined profile is split at the seam position such that the cut is parallel to global directions. When the profile upside is changed, the output length is not affected.

TR285866
When a Physical Connection type is Teeweld Y, the physical connection parameters NoseMethod is a constant, NoseOrientation is equal to 90, and RefSideFirstBevelAngle is equal to 0. These parameters are no longer being manually overridden, which is giving correct manufacturing output information (i.e. multiple bevel information on the manufacturing part output is no longer missing).

TR286008
When ‘SNU_BOX’ algorithm is used for unfolding a specific manufactured plate part, the outer contour/the geometry of the plate part is now correct in part monitor output.

TR286806
The Pin Jig boundaries list is now displayed for an assembly which has an internal seam, and has the internal part not included in the assembly.

TR287891
Users are now able to edit markings when a marking line is drawn with a constraint in sketcher.

TR287898
When user attempts to modify part with free edge treatment using the annotation editor, the end result is not a duplicated annotation.

TR288123
When Base Plane Type is set to "TrueNatural" by Rule, starting plate template command displays no error message, Template Service rule creates base plane.

TR288211
Remarking lines are now trimmed to template boundaries. **(Also implemented in 10.00.73.0088).**

TR288398
Curved Skeg plate is now generating correct XML and displays correct geometry after manufacturing in part monitor.

TR288868
When user manufactures a member having end cut with rat hole shape in it, the contour is now correct **(Also implemented in 10.01.15.0086).**

TR289210
Plate panel parts are now being aligned based on longest edge. Parts are viewed in part monitor and exported to Nestix without any angle orientation **(Also implemented in 10.00.73.0087).**

TR290017
In a specific test case of manual Physical Connection (Teeweld) in customer model, both parts have bevel nodes. Bevel nodes are now removed when its properties are zero.

TR290826
When user modifies the bevel attributes on a contour, it will not cause closed contour validation failures that cannot be resolved.

TR290828
When modifying either annotations or markings using the attribute editor, values are now not reset to their defaults after hitting apply.

TR290831
The support marks generated for knuckle marks, roll marks, and common seam marks no longer cause validation to fail.

TR290967
The name of some of the annotations in the SMS_ANNOTATION file are confusing to the end user (i.e. Margin-Fixed). These names are updated to more accurately reflect the purpose of the annotation. In addition, the SMS_VIEWER file needs to be updated so that the correct attributes are always added to newly generated marks and contours (i.e. face type needs to automatically be added to new profile contour segments).

TR290974
The unfolded Manufactured Hull plate parts now have no notches where the plate contours come to a point. Each contour has no Gap or Shift Value from the physical connection.

TR290977
When a user checks a profile part in part editor and accepts without any changes and when user re-opens part editor, the profile bevel annotation does not move.

TR293681
Part validation now passes as the properties (Bend direction, roll boundary angle, radius, etc.) of the roll boundary mark (acting as support only mark) does not change unexpectedly when a knuckled plate is edited in part editor. User can now complete the validation if the same plate is to be edited using part editor for second time.

TR294360
User creates a multi-plate panel and places manual marking lines on only one of the plates. Then user views panel in part monitor and reviews the manually place marking lines. Hovering over the lines, quick pick will show that multiple lines have now not been created equal to the number plates making up the panel. All Geometry Attribute Values are the same.

TR295636
A user modifies a detailed part which is already in the common part group and then manufactures the part. User re-runs the common part service resulting in part in a unique part group. User tries to run Manufacturing Service Manager to update to modified part, the error is now not reported “No manufacture object exists under selected object”.

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Structure

TR161174
If we select three member systems to copy and paste elsewhere in the model, the APS plates that are joining these members should not be automatically copied. This is resolved.

TR169994
A Sketched feature constrained to another sketched feature does now mirror correctly. The constraint is now applied to the right line.

TR200239
The option to choose either end of the intersection is available when placing the radial pipe members.

TR205478
Users can now successfully import slabs from Frameworks+ to Smart 3D.

TR216215
Naming rules for stair objects have been added in the delivered catalog.

TR218253
Slabs copied from an original slab placed using the Angle to Plane option are now pasted at the same elevation as the original slab.

TR234689
Users can now draw slab boundaries correctly using Sketch2D.

TR235994
A slab can be created using the sketcher with the Plane by 3 Points option, using another slab as a reference.

TR236177
When an opening is created on a member, the opening and the second-class objects (OpeningType, StructSketchWireBodyOutput, GeometricConstructionMacro, GCCoordinateSystem and GCFacePortAdaptor) have the same permission group.

TR240534
Volume by Selection can be used successfully for creating volumes for walls.

TR242480
The flange cut is now being re-symbolized correctly in the profile sketch layer.

TR251423
The ladder orientation no longer changes after modifying the location of an elevation plane.

TR251937
Graphics for specific slabs referenced in Reference 3D models are displayed correctly.

TR252866
The closed path handrail will behave correctly with respect to its Horizontal Offset settings.

TR257671
Users can now place circular openings on a circular slab without producing a To-Do List warning.

TR262556
Correct database data is reported after changing the composition of a slab.

TR263017
Built-up sections can be placed using the Place columns at Grid Intersections command.

TR263612
The location of an imported Vertical Brace from PDS is matching with the Smart 3D model.

TR264421
Users are able to move Merged Footings (By Point) from both host and satellite locations in a Global Workshare configuration.

TR266972
The face position and the boundary offset of a slab do not revert to default values after changes in the slab composition.

TR267067, TR270209
Users can split a plate when the plate is being used as a sketch reference for a slab/wall/ member opening, which is in a different permission group from the plate. **(Also implemented in 10.01.15.0070)**

TR267168
Custom attributes can now be imported from Tekla to Smart 3D by using specific bulkload and mapping files.

TR270289
Custom baseplates can now be created and copied/pasted from the catalog to a new location.

TR274540
Users can now control the location of the knuckled bracket when the support stiffener is an edge reinforcement.

TR276219
The wall weights change with respect to changing the material type of the wall system.

TR276878, TR279073, TR279074, TR279075
No crashes will be seen while importing a specific Tekla .stp file into Smart 3D. **(Also implemented in 09.01.30.0115, 10.00.73.0078, 10.01.15.0079)**

TR277270
Standard members with assembly connections will split without error.

TR277327
Users can now change the dimensions of windows on a wall using the Properties page.

TR280200, TR282565
Users can modify the thickness of a circular slab with openings. **(Also implemented in 10.01.15.0081)**

TR280378
The width and length of BoundedRectPierFootingAsm or RectSlabFootingAsm components can be modified.

TR281799
The user has ability to place GC GCFacesFromCrossSection when using Coordinate System at member frame connection.

TR284623
The CurveExtend GC is now producing correct results when the end of the input curve is near a hull patch boundary.

TR285865
The area of bulb bar cross sections is now being calculated correctly.

TR285902
Tapered members can now be exported using CIS/2.

TR287215
Plant assembly connections are selectable in Plant and Material Handling Edition model types.

TR288052
The preview file for bulb cross section available has some parameters now shown correctly.

TR288167, TR288682
The Convert to Member System command now works successfully if the member system containing the parts is placed using equipment or designed equipment as the parent system. **(Also implemented in 10.01.15.0086)**

TR288271
Members with offset values in frame connections correctly align when imported.

TR288948, TR289718
The Member Autoconnect command uses the correct beams to create vertical corner connections. **(Also implemented in 10.01.15.0086)**

TR290497
The walls at a satellite location on a customer database will now edit after transferring them from the host to the satellite.

TR291066
There are two objects, each belongs to a different location (Host and Satellite).
The user at the Satellite location cannot modify/delete an object that belongs to the Satellite because of some relationship that it has to an object at the Host. This relationship does now appear in the Relationship tab for either object, and relationship was manually created between the objects.

TR291656
An opening created on a slab has the same permission group as the slab (as it should). Now, the second-class objects of the opening (OpeningType, StructRootContour, StructPlane3D, StructPlaneGenerationAE) have the same permissions as the slab permission group.

TR293001
EndCutEdgeMappingRule does not error when running the update symbol configuration command.

TR301006
Topological point location is not getting updated with the changes made in the references until point is re-computed. (**Also implemented in 10.01.15.0096**)

TR301685
Dimensional constraints on deck are getting shifted from edge to the opening after placing an opening on the deck. This results in the sketch and plate dimensions being inaccurate.

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Systems and Specifications

TR288235
Modification of pipeline properties now triggers the automatic generation of a name according to the selected NameRule in order to be consistent with the Property Page behavior of all other object types.

TR290416
The behavior of System Property Pages has been corrected to show all property descriptors, maintain visibility of Apply button across tabs, and address name rule issues.

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Translators

TR185920
Eccentric reducers in RVM files translated using ConvertToZVF.exe are now positioned correctly in the newly-translated VUE file.

TR237488, 239369
Dimensions no longer scale by 25.4 when saving the drawing to DWG format. **(Also implemented in 09.01.30.0082)**

TR241567
Slant Cylinders (SLCylinder) can now be imported from PDMS without any graphical issues.

TR242733
Error logging is improved to specify the reason when “ConvertToZvf” fails to convert a V8 DGN file due to unavailability of MicroStation V8 on the machine.

TR243127
When running Smart 3D and SmartPlant Interop Publisher (SPIOP) on the same PC, SPIOP registry settings are no longer cleared after running an application using Smart 3D’s delivered sprb.exe application (i.e. ConvertToZvf).

TR245822, TR261221, TR261222, TR261223
Structural members are now imported with correct orientation from PDMS irrespective of hierarchy. **(Also implemented in 09.01.30.0097, 10.00.73.0061, 10.01.15.0063)**

TR256656, TR258134, TR258135
Pipe runs with Surface Mount Components can now be exported to PDMS. **(Also implemented in 10.00.73.0058, 10.01.15.0062)**

TR257615
Compartment attributes “Space_Group, Space_Type, Function_Type” are now exported to Neutral XML file

TR257788
PDS models with empty DRV files can now be attached as Reference 3D models after being translated through SmartPlant Interop Publisher (SPIOP).

TR258528, TR260361, TR260362, TR260363
Tribon ClipMould and ClipTop type of Clip objects can be imported as Smart 3D Regular Plate parts. **(Also implemented in 09.01.30.0097, 10.00.73.0060, 10.01.15.0063)**

TR258552, TR258553, TR258562, TR259555
FacePlates which are not modeled at midway of mounting face are no longer imported with incorrect dimensions. **(Also implemented in 09.01.30.0097, 10.00.73.0060, 10.01.15.0063)**

TR259286, TR259721, TR259725, TR259726
Smart 3D now supports importing of PDMS Panels as Plates. **(Also implemented in 09.01.30.0096, 10.00.73.0060 and 10.01.15.0063)**

TR259497, TR261339, TR261384, TR261385
Pipe stocks with different option code can now be exported from Smart3D to DATAL files. **(Also implemented in 09.01.30.0097, 10.00.73.0062, 10.01.15.0063)**

TR260508, TR275376
Mapping file is not processed multiple times when Importing PDMS DATAL into Smart 3D. **(Also implemented in 10.01.15.0077)**

TR260509, TR262685
DATAL files having $$ symbol in PIPE-NOTE now get imported in Smart 3D. **(Also implemented in 10.01.15.0065)**

TR260610, TR262686
When importing a DATAL file, mapping a single dimensional attribute twice in the mapping file will honor the latter value only. **(Also implemented in 10.01.15.0065)**

TR260959, TR262011, TR262012, TR262013
Pipe runs are no longer imported multiple times when there are empty Pipe runs in the DATAL files. **(Also implemented in 09.01.30.0098, 10.00.73.0062, 10.01.15.0064)**

TR260966
Error logging is improved to specify the reason, when import of Tribon XML fails due to unavailability of default material grade in the Smart3D Catalog.

TR261045
Generating mapping tool is now enhanced to generate “pipe_flg_so" attribute in the mapping files.

TR261401, TR262171, TR262172, TR262173
PDS Custom Design Equipment can now be imported into Smart 3D as G-Types. **(Also implemented in 09.01.30.0098, 10.00.73.0064, 10.01.15.0066)**

TR261677
Import no longer aborts if a stiffener is incorrect in the XML file.

TR262313, TR265472
Imported pipe runs orientations are now correct when a socket elbow is present. **(Also implemented in 10.01.15.0066)**

TR262477, TR265756
Graphics are shown correctly for Reference 3D data converted with ConvertToZvf utility. **(Also implemented in 10.01.15.0066)**

TR262908, TR266753
The custom attributes defined on Equipment can now be imported into Smart 3D with proper mapping. **(Also implemented in 10.01.15.0067)**

TR263459
Users can now map codelist numbers to short or long description of Compartment attributes to get the information when exporting to Neutral XML.

TR264103
If an assembly connection is placed between an Approved member and a Working plate, when the user is attempting to delete the plate by removal from an XML file through import, now a message stating “Error occurred while deleting the objects” will be logged into the import log file.

TR265125
Export to PDMS and Neutral Schema performance has increased by opening Excel.exe a minimum number of times during the process.

TR265223
The Import tool now imports with a custom name if no names are defined for SITE, ZONE and UDET in DATAL file.

TR265799, TR267062, TR267118, TR267179
Tribon faceplates can be imported into Smart 3D with proper offsets and dimensions. **(Also implemented in 09.01.30.0102, 10.00.73.0066, 10.01.15.0068)**

TR265886
Neutral XML with numerous hull profiles in model now exports successfully.

TR266021, TR266111, TR266112, TR266113
HVAC Mitered Elbows can now be exported to MicroStation with correct graphics. **(Also implemented in 09.01.30.0101, 10.00.73.0066, 10.01.15.0067)**

TR266701
Smart 3D now handles log file creation when the log file of same name exists at the destination folder.

TR267353, TR271617
Data is now loaded into Smart 3D successfully when updating SmartPlant Interop Publisher (SPIOP) data for Reference 3D objects using Batch update. **(Also Implemented in 10.01.15.0070)**

TR267366, TR269894
PDMS Plates, Slabs, and Openings with a negative arc radius can now be imported into Smart 3D without any errors. **(Also implemented in 10.01.15.0069)**

TR267566
Olets are now imported successfully even though two or more Olets on a pipe run have the same positional co-ordinates.

TR267945
Semi-Elliptical Shapes now export to MicroStation DGN properly.

TR268821, TR272971
PDMS Import is able to set properties of nozzles with Ring Type Joint. **(Also Implemented in 10.01.15.0072)**

TR269408, TR271443
A mating flange is no longer inheriting wrong specification at spec break location. **(Also implemented in 10.01.15.0071)**

TR269842, TR271432
A connection between pipes and nozzles is formed after being imported, even though the nozzles do not have a name or ID in the DATAL file. **(Also implemented in 10.01.15.0072)**

TR270314, TR273059
PDMS Olets and Eccentric Reducers are no longer imported with incorrect names. **(Also Implemented in 10.01.15.0077)**

TR270943
Import of PDMS Panels no longer fail to import as Slabs when MATREF does not exist in the DATAL files.

TR271045, TR274785
PDMS Import Validation Tool no longer generates additional properties which are not present in the .ATT file and it is not generating the required attributes like NAME, TYPE etc. **(Also Implemented in 10.01.15.0075)**

TR271057
Fixed a problem where import of DATAL files having CAPS and BENDS failed due to same LPOS and APOS co-ordinates.

TR271441
Spec Break situation within the run is now handled properly by PDMS Import tool.

TR271527, TR274456
All “NEW WELDS” in PDMS DATAL file can be imported into Smart 3D as Field Fitted Welds (FFW). **(Also Implemented in 10.01.15.0075)**

TR271605, TR272644
When importing PDMS piping data with mating parts having different short code and option code, no pipelines get skewed. **(Also Implemented in 10.01.15.0072)**

TR272108, TR275701
It is now possible to export names of piping components which have Cyrillic characters. **(Also Implemented in 10.01.15.0077)**

TR274349
Tripping stiffeners can now be exported to Neutral XML.

TR274447
Torus and Cone shapes under design equipment now get exported to MicroStation DGN files without any issues.

TR275368, TR280614
Smart 3D now handles import of large equipment PDMS DATAL files containing many equipment shapes. **(Also implemented in 10.01.15.0079)**

TR275413, TR277390
Users can export to network path with a dot in path address. **(Also implemented in 10.01.15.0078)**

TR275855
Custom labels created from the Catalog task can be exported to SPRDirect files.

TR275867, 277261
Panels can now be imported into Smart 3D as Slabs with default composition and type if the mapped Composition and Type are not present in the Catalog. **(Also implemented in 10.01.15.0078)**

TR276064
Eccentric Cone or Snout primitive shapes export as concentric cones, when exported from Smart 3D to PDMS. Now it is fixed by mapping X-Offset and Y-Offset in the mapping sheets.

TR279003, TR279334
ZVF files generated from AutoCAD dwg files can now be attached as Reference 3D with "Graphics Only" model type from Project Management. **(Also implemented in 10.01.15.0078)**

TR279939, TR280270
Compound filters can be used to export data via Neutral XML or GeniE **(Also implemented in 10.01.15.0079)**

TR283096
Graphics are no longer missing when a DGN file is inserted as reference.

TR288060
Planar Plates with several faces no longer fail to import into PDMS.

TR288693
Tribon Import has been improved to better handle stiffeners on curved plates.

TR289901
Repair Tribon XML tool has been improved to repair the stiffeners placed on both sides of the panel.

TR290406, 293042
Equipment properties are inherited from DATL file to S3D even though the structure data is available under the Equipment hierarchy. **(Also implemented in 10.01.15.0088)**.

TR291101
The sketch direction for opening cuts is now properly exported to PDMS.

TR291394
In a model, Local_System is now found under Structures node for Profiles when exported through Genie.

TR292365
ZVF file converted from CDW file by SPIOP is able to attach as R3D without missing graphics objects in S3D.

TR292459
PDS optional straight piping feature split by Olets is imported to S3D optional pipe properly.

TR293186
If a user does not have permission to a system folder and attempts to import TRIBON structure into that system folder, the structure is imported into the Root Ship node. This can cause issues because the TRIBON structure is not in the workspace filter and will disappear when the session is refreshed.

TR292861
If the plate system has quotes in the name, the object is processed with no errors reported in the log file, and now external file is created with name format
<<OIDoftheObject>>.<FileTypeExtension>>

TR294324, TR286206
S3D export to DGN command is able to create correct graphic now. **(Also implemented in 10.01.15.0089)**

TR295863, TR294048
Import data from PDS to S3D does not import extra pipe feature now. **(Also implemented in 10.01.15.0091)**

TR296443
Support modification for construction information properties for multiple R3D elements in select set from single R3D Project. The select set should not include objects other than R3D elements, not even R3DAttachFile, R3DFolder, and R3DProject object.

TR300149
Ability to customize S3D hierarchy while exporting to PDMS is implemented.

TR302652, TR297518
Database log file size is no longer increased exponentially when new R3D file is attached. **(Also implemented in 10.01.15.0096)**

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Weight and Center of Gravity

TR222369
Weight and CG of Piping and Equipment has been updated to better handle user defined inputs.

TR227751
Able to multi-select components using SHIFT button in Weight CG calculation.

TR260626
When selecting a user-defined coordinate system to obtain the Weight & Center of Gravity properties of an assembly, the Center of Gravity values are now calculated with respect to the user-defined coordinate system used.

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Open Problems

Setup

TR240662
A client machine that has Name Generator installed will place everything as 'Unspecified' if Name Generator is uninstalled. The user must reinstall Name Generator to remedy the problem.

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Catalog Data / Reference Data

TR87045
The Verification Utility does not verify the operator symbol when the user checks the "Verify symbol placement option".

TR110626
Piping Commodity Procurement Data should not require a NPD in the Gasket Part Data.

TR125733
If both default bolt and optional bolt are available for a bolted joint, the verification utility should create a To Do List record for the missing Nut/Washer for every valid bolt.

TR159220
If an embedded label is used for the ShortMaterialDescription, entries in the Verification Report show only the label.

TR193608
When a spec is defined in English units and the user is interested in defining a particular component in metric units in the Piping Commodity Filter, the item is not listed in the drop-down menu when trying to place the component.

TR199498
Running Bulkload twice without exiting results in errors.
Workaround: Exit Bulkload after every run.

TR205693
The European inch character displays as a "?" in some Support Part Numbers in the Catalog database.
Workaround: The user can modify the appropriate datasheet using a U.S. keyboard and re-bulkload the datasheet containing the changes.

TR211561
The Verification Utility does not report missing Service Limits for MSSQL catalog databases if some entries define Nominal Piping Diameter ranges and some do not.

TR213817
When the Piping Material Control data is configured such that a specific Valve can be selected with the same type of Valve body but can have different Valve operators, then the Valve operators are not placed correctly as per the defined data.

TR223716
TappedHoleDepth is not part of the delivered Piping Commodity Material Control Data engineering check report.

TR228037
An incorrect valve operator is placed when defined with many-to-one options on Piping Commodity Material Control Data.

TR231362
The Verification Utility should not report the washers undefined in washer selection filter if the washer creation option is disabled in the Piping Material Class Data for that specification.

TR231971
The Verification Utility does not give errors if SecondSizeUnits data is missing in the Piping Commodity Filter for the specification.

TR237773
On the weld clearance rule in the class rules of a piping spec, the field listed as “Weld Clearance Diameter Increase” should actually be “Weld Clearance Radius Increase”

TR238321
If a user misspells the catalog database name when using command line bulkload, the user will receive the error message “CheckIfGivenDBisUptoDateFailed to retrieve the Version numbers from the database".

TR238558
Users are unable to change the PadWidth and PadThickness on the R-Pad property page even though the Properties or Occurence Attributes on the ReinforcingPad part.

TR238954
Users are unable to delete existing sections from the HVAC spec, and the bulkload log file does not report whether or not the sections were deleted.

TR252041
A bulkload error occurred while loading versionable specifications into a specific Smart 3D empty catalog.

TR252677
After running the Verify Consistency Report from the Catalog Task, a customer is seeing different values for the Calculated Bolt Length in the report as opposed to the model. The model shows the correct Calculated Both Length, but the report shows an incorrect value because the report is not including the Gasket Thickness.

TR260385
The bulkload log reports that the Occurrence attributes failed to load, but the attributes are loaded to the catalog.

TR262425
The Verify Consistency Report does not list the data for which FirstSizeFrom column is defined and FirstSizeTo column is not defined in Piping Commodity Material Control Data.

TR283665
When the user bulkloads standard notes data with text values, the Catalog task does not display these entries.

TR286307
When a name rule is deleted from the Catalog task, all the classes using that name rule will also be altered.
Workaround: User can delete the name rule for these classes using Bulkload Tool.

TR288968
Smart 3D currently assigns incorrect content to paddle spacer FacetoFace and/or HandletoCenter dimensional data if the part is bulkloaded to the catalog with content missing in these parameters.

TR293052
The Wall Thickness and the Piping Inside diameter of Nozzles are not consistently shown for the ports that are defined without schedule thickness.

TR295094
On-the-fly parts are not shown in the Catalog task when a Geometric Industry Standard value is provided for the part.

TR295551
Default profile load point cannot be modified through a reference data bulkload, only through manual modification in the profile's property page.

TR300163
Reference data does not correctly process the disconnection of conduit bends, therefore causing the bend graphics to disappear and a To Do List entry to be created for the part.

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Civil

TR279451
In a specific workflow, if the user changes the trench composition in the catalog and synchronizes the model with catalog, then modifying the width of a trench run does not get updated.

TR286258
When the user modifies the trench cross section symbol file and runs synchronize model with catalog, the trench run dimensions are not updated correctly in the properties dialog.

TR291584
The properties of Trenches in non-working status are getting updated after bulkload and synchronize model with catalog.

TR292673
Trench property values are not retained for additional placements.

TR294019
Trench Cross Section is not updated during synch workspace with catalog.

TR294930
Incorrect Trench geometry after MDR.

TR301482
Unable to use some commands in CIVIL task after ‘Review MDR Results’.

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Common

TR187846
Properties that use a user-defined string have issues when the string is longer than 401 characters.

TR195924
Many orphaned Point3D objects are created by the SmartStep and SmartSketch commands.

TR196402
Claims command fails and stays in an endless loop when SmartPlant Foundation (SPF) server is unavailable.

TR212933
Large AutoCAD files fail to insert properly.

TR240662
A client machine that has Name Generator installed will no longer work as a machine without name generator until Smart 3D is reinstalled.

TR258566
End-cuts of a profile bounded by another profile are not being copied with the CommonPartCopy custom command.

TR 265258
Copy by Family modify highlighting/selecting has issues for leaf systems on S3D taskhost as well as WorkSpaceExplorer.

TR 271899
User executes Copy Similar/Symmetry to copy a plate that has been split by another object, which creates an intersection seam and 2 Logical Connections (LC). User then deletes the intersection seam on the source object and the LCs are deleted as well. When performing a delete target group copy, the intersection seam is deleted on the target, but not the LCs. A To Do List message is generated, "Unable to retrieve the split operation or the splitter in the plate graph. Please check that the connection is still valid and that the plate and its splitter are OK. Otherwise contact your support."

TR272476
Sketch3D properties page does not honor the active coordinate system.

TR272858
Collar clips gets flipped on target due to the Copy by Family command.
Workaround: Manually flip again to the correct location after Copy by Family command.

TR274898
UpdateNames command does not reflect the name of the existing occurrences in the model when the name rule ProgID is modified.
Workaround: Open the properties page of the object and select any name rule and click Apply. Now again select Default name rule and on clicking apply, the name will get updated.

TR 275453
After Copy by Family, Copy Similar is performed on a Profile by Table, the profile's Landing Curve Reference Offset value changes from positive to negative. "Error creating/modifying landing curve by table" error was seen on the To Do List when this change caused the points to be off the plate the profile was stiffening.

TR 280875
Smart plates like Brackets (created from Bracket assembly connections), Collar plates on corner features, and Bearing plate AC are not reported in Audit report.

TR 287267
A longitudinal corrugated Bulkhead/Swage Bulkhead are copied at source location for in copy similar instead of target location. Model Data Reuse copies structure as expected.

TR 287775
Knuckled bracket does not copy during first attempt. User needs to run copy again, during this process the supports (profiles, Bulkheads) are copied again (Duplicated).

TR287397
The Units of Measurement conversion for angular velocity requires a correction for cases that call on user defined interfaces scoped by this property.

TR288670
Control point with a diameter equaling zero generates invalid geometry.

TR 290521
Boundary modifications are not propagated for plate bound to plate edge case.
Workaround: Manually modify the target plate system with the required boundaries.

TR 290616
The target (copied) Geometric Construction (GC) opening is listed in To Do List with error message (missing mandatory Input) at source, if the support for GC opening is modified and Copy by Family modify is run.
Workaround: User manually modify GC opening and run Copy by Family again.

TR 294123
When performing a Copy Across Models to a Target Coordinate System at a different origin than that of the Source Coordinate System, sketched object to not transform to the target location
Workaround: Modify each sketched object and move it to its proper location on the target plate.

TR294793
Error appears while trying to place a Pipe Split at a satellite location.

TR 295598
When the by range is used for the copy across models method an error is generated if any of the coordinate systems within the model do not contain any definitions of planes in the X direction. The error message is:
'You do not have permissions to access this object'.
The message can be dismissed and the command will continue to populate the list with coordinate systems and users can continue with the command. It should be noted that this error message will appear for every coordinate system that does not have planes in the X-direction.
Workaround: User clears all messages and waits for coordinate system list to populate.

TR 295794
All objects are not getting copied in specific model when trying synchronise family created by user and then run “create “command.

TR299565
Piping objects are not visible in “Looking Forward” view when using the “Shaded with Hardware Enhanced Edge” style.

TR295888
Crashes may be experienced when trying to delete large amounts of objects in a single transaction.

TR299648
Symbol flavor identification issues may cause Solid Edge objects copied with MDR to display incomplete graphics.

TR300688
Default colors changed & Apply default colors throws error in a particular customer DB.

TR 300777
Copy by Family modify does not propagate if there is In-Line and End can together on the Tube member.

TR301052
Property filter for members is not working in specific case in drawings.

TR301416
Model locate does not work for non-group R3DElements. Due to this limitation, Smartsketch3D options are not shown for some of the objects in R3D Model.
Workaround - Switch on “Silhouette Edge” constraint and switch off “Point on Surface” to be able to locate the edges on translated PDMS shapes.

TR 302341
If there are changes in the type of Assembly connections for Standard member after Copy by Family operations, then Copy by Family does not pick the assembly connections as the candidates for update and assembly connections are not modified at target.

TR 302431
Copy by Family Batch does not work when Batch Manager Mapping is set to one user and another user is logged in (i.e. Batch Manager mapping is set to User1, yet User2 is currently logged in). User2 will receive an error regarding the log file and nothing get copied.

TR 302465
In Copy Across Model, Audit report reports incorrectly the number of assembly connections for member connected to plate part.

TR302760
Common Move command have inconsistent behavior if includes read only items. Smart3D has an inconsistent behavior if moving a set of objects, where some of them are in PG where user has no write access or object is non-working state.

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Common Route

TR179466
Users are unable to set the property "IsSpoolOversized" on penetration spools in the properties page.

TR209774
Underground/Sanitary specification has piping data for fittings with end preparations Spigot end x Bell end and Bell end x Bell end. However, when placing bell end reducers on spigot end pipe the pipe end will be converted automatically to Bell end.

TR260328
Insulation is not added correctly to features in a run that previously had insulation applied and removed.

TR268552
Pasting a run from the catalog does not change the run name if the user is pasting the run under the same parent pipeline.

TR269758
The access controls (like View Access and Common View) do not honor the change in active coordinate system while actively routing with the Remember Working Plane option set to true.
Workaround: Terminate the routing command, change the active coordinate system, and then begin routing again.

TR288405
When routing with a specification that has an optional pipe schedule, the mating flange logic is inconsistent when the commodity filter has “match” listed as the schedule.

TR292727
When moving a piece of equipment connected to a pipe run, the run does not disconnect from equipment and a To Do List entry is generated.

TR292926
When using Quick Route to route between two nozzles correlated to a P&ID, the pipe run name is not propagated from the P&ID.

TR302486
Route of multiple, consecutive straight features in an invalid branch throws Application Error after writing to the To Do List.

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Compartmentation

TR274291
If a Linear Extruded Plate or Non-Linear Extruded Plate is used and there is a sketched boundary creating an overlap on one of the adjacent faces, the compartment will fail to generate.
Workaround: If a sketched seam is used in lieu of a boundary, the compartment generates as expected. Angle does not matter.

TR284908
User update attributes of a compartment of type compartment may take a while in a model where many coordinate systems with many planes exist.

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SmartSketch Drawing Editor

TR189177
Centroids of filled object sometimes fall outside of the fill range. At this time, it is not known what causes this to happen.

TR247762
When translating SmartSketch *.sha files to MicroStation *.dgn format, the Smart Frame Display Manager settings are not honored, and default values are used.

TR264926
Background color of the drawing document changes to its 'Default' upon doing some changes.
Workaround: This is dependent on the user. If the background color on the user's machine is set to 'default' then we see this problem. If it is set to some other color (ex: Black), then we don't see this issue.

TR295637
When a drawing is saved to Auto CAD 2007 (dwg/dxf) format, its corresponding reference files are getting generated with higher version of AutoCAD (i.e., AutoCAD2010).
Workaround: Use Merge References = 1 in the itacad.ini file to generate a single file.

TR297924
Text formatting from Properties → Notes tab, is not getting honored when the text is copied and pasted on the drawing sheet after updating the drawing.

TR303107
Angular chain dimensions are not translated correctly when saved as DGN.

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Drawings

TR143497
When modifying a section, the user is allowed to select another section view. This causes modifications made to the first view to not be committed properly, and that is not necessarily noticeable to the user.

TR143498
Modifying a section view height can cause automatically placed labels to shift location.

TR163553
SAT files are only partially complete when processing large MicroStation design files. The processing ends with a successful status, so there is no indication of an error.

TR189497
Dimensions placed or moved after the scaled sketch dialog bar is invoked are not remembered.
Workaround: When scaled sketch dialog is invoked, user must select a smart frame and Finish the command. Do not manipulate dimensions after invoking scaled sketch.

TR192482
Revisions and Issues are not ordered when returned from a COM label. Therefore, they are not useful for reporting.
Workaround: Write a Visual Basic label/report.

TR195934
Control points added programmatically to supports will get labeled in a support drawing, but control points added manually will not get labeled.

TR200791
SAT files which have been imported to a model with the Equipment task and then exported to MicroStation using the 3D DGN component sometimes do not display correctly in MicroStation.

TR200920
The volume size created for Drawings by Query drawings is too large if the focus element has a local coordinate system different from global and if the SytemRangeNavigator module is specified for navigating rule.

TR214473
HVAC and pipe straight features do not re-symbolize if the midpoint of the feature is not included in the volume.

TR225861
Unable to apply line styles to inserted 2D MicroStation design file objects. The default styles will apply (such as Dash, Dash Dot etc.) but others, like ISO Narrow Dashed, will not work.

TR226039
Section views that are created on drawings which have a viewing direction relative to the “Object CS”, and that coordinate system is different from global, can generate oversized volumes and therefore unexpected drawing sizes.

TR229596
When a package is saved in the Drawings and Reports task there is no feedback that is has been saved. If a user does not have permission to write to the symbol share the package does not get saved but the user is not aware of this.

TR242001
A Manufacturing Profile Sketch drawing will also pull in member parts contained in the assembly and then stall when a drawing update is attempted. This results in incorrect items gathered for the drawings. When updating the drawing, it stalls when a member part is reached forcing the user to “abort” and restart the software.

TR242238
Cannot edit Layout Templates if the “\” character is at the end of the Shared Content Path in Project Management. Either a Method of Object Failed or Illegal Function Call errors will be displayed.

TR243469
Drawings fail to update if C:\Temp does not exist on the user's machine. No ship drawing views will update for a user if C:\Temp does not exist. The error log only states "Error: Path not found" with no indication of what path it is looking for. This makes it very difficult to find out why their views won't update when all that is missing is a folder on the C:\ drive.
Workaround: Create the C:\Temp directory. This could be problematic if a user is not an administrator on their own machine, or in a more likely scenario, if a drawing is being submitted to a batch server where C:\Temp doesn't exist.

TR245390
Drawings have missing graphics if the volume used to create the drawing does not include the Coordinate System of designed equipment. This only applies to designed equipment which has imported *.sat files.

TR246209
The anchor dimensions between Plant type objects are not working in Drawings by Rule drawings. Anchor Dimensions, the dimension between different types of the objects (i.e. between equipment and a member) is not working in Drawing by Rule drawings.

TR246697
Users can make edits inside a drawing view by double clicking on the view border. For composed drawings, the Scaled Sketching button on the toolbar is available when this is done. If the scaled sketching is turned off all other buttons become enabled. This allows users to place items, such as views and labels, which should not be allowed inside of the view. This can result in various problems including causing draft.exe to hang.

TR253200
Drawing labels cannot be placed when a label is not in a root folder. This prevents users from creating subfolder under the root folder and moving labels to them.
Workaround: Moving labels to subfolders will work as long as the *.xsd files are copied to each subfolder. Secondly it is necessary to have a single drawing label xml file in the root directory. This label will not place but is required to be there, it should be a copy of a label which can then be renamed to something like, "NonPlacingLabel.xml."

TR259823
If a volume drawing components is in 'InReview' status, it still allows the users to create volumes using these components, but it prevents creating drawings for these volumes even after changing the volume drawing components to 'Working' status. This results in unnecessary volumes in the model.

TR260396
The cutting plane arrows will disappear if the view label rule is not defined.

TR260635
'DrawingPGControlPoint' Point Generator module is not considering the control points as precedence for Design Equipment objects. Instead it is pointing to the center of that Design Equipment objects range. The dimensioning and labeling with this point generator always points to the center of the design equipment range even though they have the control points.

TR260964
The drawings from Reference3D objects may fail to update because of 'Polymesh' objects especially when these objects are clipped by the drawing volume.

TR262121
The background color of the drawing document changes to its ‘Default’ upon doing some changes.
Workaround: This is dependent on user. If the background color on user’s machine is set to ‘default’ then we see this problem. If it is set to some other color (ex: Black), then we don’t see this issue.

TR261432
In an Oracle database, when making a Steel Order (Shell Profiles) drawing the non-target hull plates are drawn incorrectly.

TR262440
‘Reinforcing Pad’ graphics on 2D Drawings are not showing the true representation.

TR263494
When selecting "MfgPinJigRuleset.Default" with "Assembly Drawings" from Define View Styles, it crashes.

TR265503
Users are unable to print the drawings from Drawings & Reports Environment using ‘Right Click → Print’ command, if the drawing document name contains special character double - quote (“).

TR266472
Hidden lines are getting drawn even when set to <Not Drawn> when using the “PlateSystemDrawingWrapperEntity” or “PlateOuterContourDrawingWrapperEntity” drawing wrappers.
Workaround: Manually modify the drawings.

TR266583
When using a custom fill style in the Graphic Rule, it will make hidden lines process differently than if you use a standard fill style.

TR266757
When Roll is applied on the entire profile, the roll radius label is not being displayed in the profile sketch drawings.

TR267064
When a comma is being used as the delimiter instead of a decimal, the auto dimensions for a Manufactured Profile Sketch are extended infinitely.

TR267375
Steel Order Expansion drawings need to display both anti-molded and molded lines.

TR267872
The dimensions of a linear extruded plate on a Steel Order Expansion Drawing are incorrect if the block or volume is not set when using the Associate Object to View command.
Workaround: Select a block or volume in the last smart step (Orient View) in the Associate Objects to View command.

TR267874
Special Characters in the drawing document names create problems with creation and update. This is applicable to all drawing types (Composed, OrthoDBQ, DBR etc.). For Isometrics, it supports the double-quote (“) but fails to print as explained in the TR265503. Some of those special characters are listed below.

- ' (single-quote)
- " (double-quote)
- / (slash)
- \ (backslash)
- : (colon)
- ? (question mark)
- < (less-than)
- (greater-than)
- | (vertical bar)

TR269329
Curved members are not being drawn in Steel Order drawings even though the gathering rule shows the object has been found.

TR269598
A drawing fails to update with deleted Reference 3D ZVF files. If a Reference 3D ZVF file is added in a model and later the file gets deleted but is not removed from Project Management, no drawings referencing any ZVF file will update later.
Workaround: Go into Project Management and delete the referenced ZVF files.

TR269713
Isometric drawing names which start with characters ‘ISO’ fail to save the ‘Spool Information’ or ‘Material Sheet Identification’ files to disk, when saved using the ‘Save As’ option.

TR269918
Hangers & Supports Drawing by Query is not reporting the ‘Trench Parts’ if they are modeled as supporting structure.

TR271153
When a user modifies a grid location manually on a drawing and then changes the view orientation in the properties page, the grid line will not update correctly.
Workaround: The user has to delete and recreate the view.

TR271386
In an Orthographic drawing, if user uses StructNamedSurfaceGeometryWrapperEntity.dll for a plate part, the incorrect side of the plate is drawn.

TR271501
The thin volume fails for corrugated plates when the view orientation, scantlings for Plate/Profile is used.
Workaround: Users can use one of the other view orientations.

TR273013
The user is unable to produce a hull expansion drawing of the Port or Starboard side of the hull without splitting it into smaller blocks.
Workaround: Output format names have to be changed so that they do not match with any other output format name subsets.

TR273491
A Drawings by Rule drawing that contains an isometric view of curved members has incorrect curved lines.

TR274823
Cable Tray ‘Maintenance’ aspect graphics are missing in the drawings.

TR276779
The leader lines point to the center of the objects if ‘GroupClippedPoints’ option is set to ‘True’ especially when the ‘DwgMatchlineWithDim’ leader module is used.

TR280074
Users need to be able to create a custom label which will fetch out details from the model properties and put the information in the drawing upon updating it.

TR280600
When a user creates an expanded or unfolded drawing of a revolved plate, the drawings are incorrect when using the ‘Both’ expansion direction.

TR280701
There are no error messages logged when an input is missing for drawing generation.

TR283466
Users are unable to sort the labels under ‘Place Label’ drop down list if the SharedContent is configured on a distributed file system.

TR285520
If the user edits a wrapper xml and changes the .dll source reference to a .dll that does not exist, then the object that is drawn defaults to being drawn with pure VHL and no warnings are given.

TR285814
Delivered Piping Plan Style 2 view styles results in duplicate labels, one for pipe and one for feature.
Workaround: Use this delivered style 2 as reference, customize your own view style by removing unwanted rows which resolves this issue.

TR286156
Curved profiles on curved plates fail to draw correctly on expansion drawings.

TR286330
A profile is not drawn as hidden line style when it is lapped with a normal plate.

TR287607
Double right clicking on a drawing document allows for opening the same drawing twice.

TR288806
The user is unable to generate template set drawings (normal and full scale) for manually added box templates.

TR289843
When user adds a text label in the inner document, it is not retained in the drawing after updates.

TR293664
‘Bring to Front’ setting on the manually placed text drawn in outer document are not respected after update of the drawing.
TR293694
Custom tests on space object type is not working in steel order ruleset. Users want to draw only selected piping parts and selected volume objects in the drawing.

TR293865
‘Polish’ special characters are not getting shown on the drawing border when the drawing is updated through batch.

TR294946
Visibility of Non-Target plates are wrong in deck plan drawing. Visibilities of Non-Tight transversal plates are wrong in a deck plan view drawing for the existing plates using steel order (Decks) view style.

TR296103
In a specific model, there is a knuckled plate with an edge reinforcement. The edge reinforcement is split at the knuckle. When viewing the edge reinforcement's profile sketches, the miter angles do not match what is output in the model.

TR296453
If a Coordinate System name has invalid characters (like a degree symbol) it prevents the drawing properties dialog to open.

TR296646
Drawing view shifts when the view is cropped and results in error when cropped multiple times.

TR297491
Performance issues and incorrect graphics while updating drawings which has .Net Handrail symbols.

TR297579
When a user places a section view of a Drawing by Rule drawing, all structure is drawn from the section volume, even if the structure was not included in the original view. For example, if a view is created that only shows objects in one particular assembly, leaving out other adjacent objects, a section view will include the adjacent objects.

TR297668
DBQ Snap-in fails to populate when a drawing name has ASCII character 31.

TR298189
Insert dimension in a dimension group fails when the dimension group is placed initially and later on the 'DefaultGraphics' wrapper is applied using the view style.
Workaround: Delete all the existing dimensions, apply default graphics wrapper and the update the drawing. Then place new dimensions which gets retained and user can insert dimensions in an existing group.

TR299827
Unable to see 'Published' Date on the approved documents.

TR300334
When a user creates a steel order (Deck) drawing with landing curve with end cuts as graphic rule type, profile is missing in deck drawing but same can be seen with Resymbolized by Rule Set.

TR302168
Drawing is flipped when user manually associates the objects in steel order shell expansion drawing.

TR302357
Composed graphic views placed inside a region are not resizing to match the layout area.

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Electrical

TR192219
Synchronize Model with Catalog command does not create To Do List entries for cables with part definition deleted from the catalog.

TR241525
Cable length decreases when a turn feature is modified to a miter.

TR270953
The dry weight of cable with part number 1x1.5mm2 – 0.6/1kV PVC 70 ARM Cu is reported incorrectly.

TR287646
The Delay Cable Update command is not skipping cable updates when cabletrays are modified in a specific workflow.

TR295369
Error Message is shown: "This is Bad Cable. It is advisable to delete"

TR296130
Copy and paste of Cable trays flip them in a particular workflow.

TR300679
Cable doesn’t pass through a Barrier run which was updated from closed CT N/W.

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Equipment and Furnishings

TR184988
A newly bulkloaded shape is not visible in a saved session file. However, it is visible when the user creates a new session file.

TR192917
After applying a constraint on a designed equipment using one of the child shapes, the constraint will not be removed if the child shape is moved to another designed equipment

TR203362
A filter based on insulation attributes of pipe nozzles returns nothing

TR299769
When locating parts of a specific design solid object in a customer dataset, S3D crashes.

TR302876
No custom shapes can be seen when placed under a designed solid. They can be seen when placed under a designed equipment.

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Geometry Analysis and Repair

TR294620
When using File > Import ACIS, Smart3D can exit if the filename for the .sat file is very long. In testing, it was found that filenames, including the entire file path, around ~240 characters are failing.
Workaround: Use a shorter path and file name for import.

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Grids

TR288157
Users cannot edit a previously created coordinate system using Grid Wizard in an Oracle global workshare configuration. The X and Y grid planes do not display in the Grid Wizard.

TR292740
Radial grids fail to place when elevation planes are above 500 meters.

TR293822
Grids will not export from customer database.

TR294785
Distorted graphics for a member rotated at angle created 1km from Global CS.

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Hangers and Supports

TR229386
High Severity Database Integrity entries for FINL Assemblies fail to clean with the Clean Database command.

TR247500
The Toggle Face Position functionality should behave the same for the supporting structure, whether it is a hanger, beam, or a structural member.

TR294659
Unable to place HS_Lisega2010 Type49 support parts.

TR296689
Unable to place specific Generic Duct Assembly Supports on some HVAC Components.

TR297520
Remove Pipe OD as an Occurrence attribute from all pipe attachment parts.

TR302718
.Net API – Incorrect Orientation of the Support Assembly.

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Hole Management

TR259112
When a user places a hole trace and tries to get the diameter to report, the field is blank but the value is reported in the properties page.

TR259493
Sketching a hole trace does not work properly when placed on hull.

TR284633
A hole trace can become out of date on To Do List and even though the cut is still correct on the plate, the output in Part Monitor is incorrect. Hole cut is missing from Part Monitor, potentially causing steel to be cut with missing holes.

TR287093
When the user copies a hole cut alone on a standalone plate and then tries to paste it, the user is unable to select the standalone plate as the input in paste dialog box. When the user copies/pastes an outfitting object, a hole cut, a hole trace and a standalone plate all together, the user gets an invalid To Do List item and an error mark is seen in the graphics.

TR288358
When the user selects piping and equipment as outfitting objects for placing a hole trace, the hole trace is created only around the equipment. However, both piping and equipment are listed as outfitting objects in the property page of the hole trace.

TR292436
When the user performs Model Data Reuse copy on a standalone plate which has hole cuts, bad traces are created and errors are reported in the log.

TR293091
When an edge reinforcement is used as the input in the Multiple Holes command, the command displays the error "Either proper intersections are not retrieved or Hole traces have been placed already."

TR293094
Holes cannot be cut on "extended" stiffener surfaces, such as those seen at a convex knuckle. Traces can be placed at these locations without errors.

TR292436
When the user performs Model Data Reuse copy on a standalone plate which has hole cuts, bad traces are created and errors are reported in the log.

TR302120
Users cannot place a hole trace for outfitting object which has the Equipment Hole Aspect without administrator rights on SM3D client machine.
Workaround: User can be given Full control for Everyone/Non-admin Users who use Hole Management in Local Machine Registry Keys to place a hole trace.

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HVAC

TR243961
The Duct Transition features formed during insertion of a HVAC part do not honor the specification-driven values of the primary duct run.

TR263948
The wet weight for HVAC inline components is computed incorrectly as the sum of wet weight and dry weight, instead of the sum of dry weight and fluid weight.

TR292691
Duct feature gets corrupt in a specific workflow when split moved onto non-working feature.

TR302502
Inconsistent behavior on connecting rotated Air Filter and Duct using common move.

TR302552
Unable to move a header SF with branch routed from SMC and branches interconnect.

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Interference Checking

TR173382
Local IFC does not clear Point Cloud highlighted interferences, even if Local IFC is disabled by the user. The highlights stay after the user re-routes the interfering object

TR277753
Dangling Reference 3D (R3D) objects are not detected by IFC. When R3D projects are modified such that new R3D files have been added, modified, or deleted (and if R3D elements in modified R3D file have been added, modified or deleted) then interferences with deleted R3D files (or elements) have been hanging around. IFC is not deleting these interferences since IFC does not know anything about deleted R3D files. When R3D project has been deleted or changed property to Non-Participate in IFC then all these interferences will go away.

TR293255
Database markers between the S3D objects and LFM cloud are not removed even after the pOint Cloud is removed from the Model.

TR302151
Duplicate Interference markers are generated when the existing R3D objects are over written with the newer ZVF files.

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Materials Handling

TR250227
The Position of Chute shapes are incorrect in the Sketch 2D Side Elevation Layout sheet for conveyors created on rotated coordinate systems.

TR247373
If the user selects a pulley assembly as a projection item in the Place Belt Components command, then different parts of the pulley assembly are shown at different locations.

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Model Data Reuse / Model Data Transform

TR182160
If a user performs an MDR transform by 270 degrees of a Cabletray configuration containing a Vertical Tee Up, the Vertical Tee Up is changed into a Horizontal Tee and the Cabletray branch is distorted.

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Molded Forms

TR225122
When the landing curve of a stiffener ends at knuckle on it parent plate, a split knuckle is generated, but this knuckle is not required. If the stiffeners are split using this knuckle, To Do List errors are seen.

TR239802
When a user creates a profile on a curved Hull by Profile Table command with 2 points and an angle as an input, the user sees a different output for -10 and 10 degree angles with respect to the horizontal. The user expects profile to be created just opposite (in case of 10-degree angle) to the -10 degree output.

TR240692
In the model, the process goes into a loop when trying to change the type of a Knuckle Reference curve in Plate Knuckle Properties page and finally gives an error message “Quit Procedure/Abort Application: Insufficient memory is available on this computer for S3D to continue the command”.

TR242712
Users are unable to bound a transverse bulkhead to a deck composed of one Linear Extruded Plate and two Non-Linear Extruded Plates; these three plates are not mutually bounded. When a user tries bounding to the three plates, they place extra sketches to form notches under the deck and avoid the gaps formed by the edges of the various plates. The end result is the creation of a plate that has connections to all three plates, but which forms on the outside of the intended boundary.

TR245702
A profile is bound to the hull and has correct connections until the profile is split. After the split, connections on one side of the split are missing. This is not true on all profiles on hull. These profiles were created the same, but have different angles under 'Orientation Rule Assignment'. When the angle is set the same, the problem does still exist, so this doesn't fix the error.

TR258993
Users are unable to create a plate with a sketch having tangential constraints.

TR260139
Users cannot create a plate bounded to a member system with an Advanced Plate System plate. The system does not provide a valid ambiguity.

TR260758
When placing an Edge Reinforcement (ER) on the remaining edge (not on the edge made of sketched boundary) using the Bound by Point command, the boundary points selection can be done anywhere in the graphic view without letting the user choose points on the edge. Users cannot select proper points to place the ER. Users cannot place an ER specifically to any small portion. In contrast, arbitrarily some ER is being placed on the edge.

TR262136
Assembly Connection and Physical Connections fail to propagate after splitting an edge reinforcement due to connected leaf systems not touching each other.

TR263205
Brackets fail to place or have incorrect orientation when the edge reinforcement is not centered. **Workaround**: The user can manually adjust the bracket tripping rule because it currently assumes the edge reinforcement is centered.

TR266347
When a user edits the intersection (Logical Connection) offset, the system does not apply the offset and error message is seen in the log file

TR268783
After a hull swap operation, several hull seams are incorrect. Seams can be rebound and ambiguity solved but the parts will remain undetailed.

TR277041
When a user tries to delete an opening on a particular plate, it gives an error message stating “Cannot delete object(s) because of unexpected error. The user also cannot modify the opening or create a new opening on the plate.”

TR277199
If a user wants to add equipment to the hull's boundary list, this will create the required cutouts in the hull. The port side bounds as expected, but when trying to add the starboard side, ambiguity is called and the user cannot finish the command.

TR277283
Split Root Systems of a plate (support of bracket) cause detailed brackets to lose their assembly and physical connections.

TR277292
Split Root Systems of a detailed plate system (split by detailed profile system) cause plates to lose their associated slots and collars.

TR278176
When a user is trying to modify a linear extruded plate by adding a sketch boundary, the system asks for ambiguity. After selecting the required patches, an error message box is displayed stating "Method of object failed". Hence, the user is unable to modify the boundary with sketch as one of its boundary.

TR280822
Hull is lost as boundary for plates after split root operation.

TR282551
The GCLineFromCs shows twice the value entered in the length field when the option is "centered".

TR282845
A profile, stiffening a deck, is penetrating a bulkhead which is split by the deck. Split Root System is performed on the deck, parallel to the bulkhead. The Collar goes on the To Do List with a “Symbol failed” error message after Split Root System is performed.

TR287278
Copy by Family symmetry modify will not work properly for a corrugated bulkhead which is created using a coordinate system created by geometric construction and a plane created by geometric construction.

TR287598
Copy mirror fails for Geometric Construction PolylineBy4Segments.

TR290174
When a user places a curved profile, and bounds another profile to the inside of the curved profile, detailing fails to correctly trim the profile and extends past the designated boundary of the profile.

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Piping

TR138593
When a Standard Support is converted into a Designed Support, by adding a part or when the Drop Standard command has been used on a Standard Support, the Support Components are not included when generating Spools with the option set to “All Shop Parts”.

TR178391
In a specific workflow, To Do List entries are generated after automatically inserting Elbows even when the ports are aligned within an acceptable tolerance from the Port Alignment rule.
Workaround: Insert Elbows using ‘Insert Component’ command manually.

TR226205
After deleting piping generic data, no To Do List record is generated on existing pipe parts.

TR226338
A To Do List record is not generated when the user modifies pipe run slope to violate the Minimum slope.

TR228944
In a specific workflow, Group Pipe Parts with query and control point generates WBS items incorrectly.

TR236791
A To Do List record is not generated in the model if the bolt data is missing in the catalog at the specification break.

TR243299
After deleting gaskets from the catalog for clamped pipe, the Synchronize Model with Catalog command does not delete the gaskets from the model.

TR267210
If the user resizes the pipe feature, the old NPD can still be selected to route from and can cause issues.
Workaround: Change the NPD at the pipe run level, not the feature level.

TR271392
When spooling the top-level block, no errors are reported in the log file for the spools that don't get created.

TR272635
Upon initial placement of an On the Fly instrument the naming rule cannot be changed to a customize name rule.
Workaround: Place the instrument using the default name rule, then after placement, open the properties page and change it to the custom name rule.

TR272636
Spool generation at a union is breaking the pipeline into 3 spools – at the union and the pipe union joint.
Workaround: One workaround is to insert a control point of subtype 'Spool Break,' and then generate the spool with the option Spool Break by Control Point as 'Break only at Control Point'.

TR282165
Users are unable to modify the Material Control Data for multiple On the Fly instruments at a time from the properties page.

TR282621
Mirror copy of a gate valve on a run with a defined flow direction causes two different flow directions after paste.

TR286936
A 0mm pipe feature is generated when moving an elbow to connect with another elbow causing a minimum pipe length To Do List record. This only happens when the bend radii for the elbows are different.

TR291404
When pasting a branch loop with two tees on another header, only one tee is pasted onto the header correctly.
TR292382
Components are not re-computing after changing the run during insertion, and then an application error is displayed when changing the flip or reference positions after the run change.

TR293424
When selecting an override string in the part override form, the compute graphics are not displayed.

TR297936
When copy and pasting a pressure relief valve at the end of a pipe, the valve is displayed incorrectly.
Workaround: Insert the pressure relief valve directly instead of copying and pasting it.

TR299252
Even though bolt length is increased after adding supplementary nuts, the information regarding the supplementary nuts is not shown on the property page or on the Isometric drawings.

TR302092
Updating the To Do Record on a correlated pipeline on the host causes To Do Records on the satellite, and upon update of the To Do List on satellite the To Do Record on the host reappears.

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Piping Isometric Drawings

TR165005
When the SmartPlant Schema Component is removed, the user receives an error when attempting to view the Extraction Data on an Isometric.
Workaround: Register the “ssplt30.ocx” file located at *[Smart_3D_Installation_Directory]\3D\CommonApp\3rdParty\Bin*.

TR190444
When the user models using size dependent Client Commodity Code data, the same Commodity Code appears for the parts of different sizes in an Isometric Drawing.
Workaround: (1) Map labels to use the “Second Code” as the primary code or (2) Create labels such that the description includes size information.

TR234913
When a Pipeline is routed at 45 degrees in plan view and then the elbow is tapped with a dummy support, the extracted isometric for the pipeline fails.

TR260112
Isometric drawings fail to generate if the StartTimeout option in the preprocessing program is set to more than 80 seconds.

TR277582
Isometric drawings fail to extract when the Bentley Plant space application is installed on the machine.

TR279799
The Surface Mounted Component (SMC) is not shown on the isometric drawing when pipe and SMC are assigned to separate block assemblies.

TR283219
The position of the support is incorrect when a design support is placed on the Y Strainer.

TR284237
The Isometric shows the incorrect direction for the Angle valve stem when the valve is on a 45deg inclined plane.

TR288747
In a specific case, the bolts are not reported in the neutral file when the gasket requirement is set as not required in the Catalog task.

TR290070
The position of the half coupling and respective quantity in MTO is incorrect if a pipeline is branched on the jacketed piping to form a half coupling.

TR290600
The Isometric do not extract for the large radius curved pipe, on specifying a value other than zero in curve threshold option.

TR293934
The Isometric drawing displays redundant NPD information for an ‘Olet type extended body gate valve’ is placed in the pipeline.

TR295469
Field welds are reported as Shop Welds on Jacketed Pipe runs in isometrics.

TR295544
Dotted Un-dimensioned filter/label doesn’t work correctly for jacketed piping isometrics.

TR296394
Support dimension is missing if support is placed on welds
Workaround: Change the dimension style to composite from basic.

TR297010
Incorrect ‘Face to Center’ dimensions appear on isometrics for threaded fittings.

TR297155
Isometric fails with disconnection message if the delivered ‘RPAD’ key is used for RPAD which is at 45-degree angle.

TR298494
Curved pipe isometrics fails when routing from all the branches placed on it.

TR299446
Incorrect isometrics for jacketed pipeline especially when the ‘DrawingRotationAngle’ value used is other than ‘0’.

TR300666
Incorrect secondary orientation is showing up on isometrics for ‘Y-Type Globe Valve’ with ‘Operator351’.

TR303464
Some of the rows under ‘Supplementary.NeutralFile.NeutralFile’ are getting deleted when trying to add new rows. This happens when many rows are included, for example greater than 200.

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Planning

TR274565
For large models, there are issues sequencing B0 in Auto-Nested mode.

TR289739
Light parts (undetailed) are being processed by Common Part Service and corresponding Groups are created.

TR292493
Profiles detailed at different times are not found as common parts.

TR292774
Common Part Group icon is going out of date when the part property is modified even when the Common Part Group is locked in the properties page and group icon changes to out of date.

TR301986
When running Common Parts > Tools > Common Part > Common Part Utilities > Test Rules on collars, the collar thickness value comes blank/no value is being reported. User is provided with no results. These results could lead to incorrect manufacturing parts to be grouped and fabricated with wrong thickness.

TR302207
User can select invalid comparison type for candidate selection type in Rule settings dialog. This could result in incorrect workflows allowed with the user interface.

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Project Management

TR166351
Editing problems exist in the Grid View of Reference 3D models attached to a model in Project Management.

TR177850
When a V8 dgn file is attached using Reference 3D, the original line styles (dash-dot, dotted, etc.) used in the V8 dgn are not preserved. However, the line styles are displayed correctly when the same file is saved as a V7 dgn file and is inserted in Smart 3D using the Insert file command.

TR186200
Not all locations are available in Project Management for the Duplicate Plant for Workshare command when a second global workshare configuration (GWC) is added to a site that already has a GWC.

TR202964
Custom SQL Query reports do not work after converting the Databases from MSSQL to Oracle or vice-versa.
Workaround: Manually edit the rqe files and copy the custom SQL queries at both "<SQL>Custom SQL query</SQL>" and "<ORASQL>Custom SQL query</ORASQL>"

TR215466
Database ownership is not transferred from a host site on a trusted domain to a satellite site on a non-trusted domain in a global workshare configuration.

TR235459
The Restore Plant as Copy command does not prevent restoring the V2011 backups if the user modifies the backup configuration file (bcf) accordingly. The direct upgrade of a dataset prior to V2011 is not supported.

TR297568
New delivered HotFix induced problem that user cannot delete attached R3D models.

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Reports

TR158692
Conditional Labels using Post-Formatted Unit of Measure (UOM) values will not work correctly when the option to display the UOM is set to ON.

TR161686
The 'Neutral File Information' label is not listing Reportable Parts in the neutral file generated after running an Isometric.

TR162070
The 'Piping Isometric Spool Continuation' label returns incorrect information on Spool Isometrics.

TR175747
If a user deletes a folder from the Copy to Catalog dialog in Drawings and Reports without deleting the Report Templates with it, several Database Integrity issues will be created.

TR200659
Differential report does not highlight the modification when a Smart 3D object dry weight value is changed from "undefined" to any value.

TR206601
Oracle function 'RPTALLCHILDRENINHIERARCHYBYOID' delivered in [*Smart_3D_Installation_Directory*]*\Reports\Tools\Oracle\ORASpecialViews.sql* is giving incorrect results.

TR212026
The Aspects are not shown correctly in the Interference Checking Report in some cases.

TR238338
Reports fail to run at Satellite locations if the Reports are created in the Host location whose Shared content path has a trailing backslash (\) at the end.
Workaround: Give the trailing backslash for the Shared content at the Satellite location.

TR239624
The Verify consistency report gives inconsistent results when the user runs the report from the Reports environment and when the user runs it from the Catalog task environment.
Workaround: Run the report only from the Catalog task.

TR242280
The Tooltips and the SPRDirect reports fail to report the coordinate system name when the user specifies to get this value from the Matrix tag of the. rfp file.
Workaround: User must select the correct coordinate system in the pin point toolbar and use CanInherit = Yes to see the desired value in the tooltip. Similarly, the user must select the coordinate system for every SPRDirect report to get the co-ordinates with respect to the desired coordinate system.

TR261758
When a report is updated via batch it doesn't go into Locked State. This allows a different user to update that report at the same time. However, this is not the case with drawings.

TR262939
Deleting a label from the catalog user interface deletes the entire parent folder. This includes the deletion of other files and folders under that parent folder.

TR263332
Users cannot run a report on Reference 3D Objects using a filter. The interface names for some Reference 3D Objects has changed, causing the delivered catalog object type filters to be in a bad state.
Workaround: Modify object type filters so they will work properly.

TR267452
Delivered MTO reports based on WBS will not respect branch reporting ownership rule.

TR280280
Unable to copy content to the clipboard while running a report.

TR294732
Manufacturing Material List Summary reports ran on Oracle display an “X” for unknown Dimension values, as opposed to “unknown” as displayed in reports ran on SQL.

TR300237
Report results are not filtered by the “Begins with” condition in Property filters.

TR300860
The sample source code provided in the delivered shared content for To Do List reports is need of an update.

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SmartPlant Enterprise® Integration

TR214277
Along Leg Feature (Branch Parent) does not get correlated when moving features into the correlated run.

TR227296
When merging an object during retrieve in the design basis, Smart 3D is trying to validate all existing properties values of the objects to merge even if these properties are not used in Smart 3D objects.

TR232728
Processing retrieved Cable Schedule fails when cable termination is in Cyrillic.
Workaround: Use names with just Latin characters.

TR234319
Changing sessions with P&ID Viewer open causes crash.

TR273272
Incorrect flow direction is shown when using One-To-Many Correlation.

TR273710
Users can modify SmartPlant Foundation registration without permissions on model.

TR289569
Object variable error when trying to re-correlate an object with design basis delete.

TR291825
Consolidated Val exists in SPF for objects that have been deleted.

TR295334
Output is missing interface when update with multiple chunks that share OwnsDistributionConnection relation.

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Space Management

TR295858
Merged Drawing space volumes of different cross sections fails to copy/paste.

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Structural Analysis

TR296925
Session file becomes unusable from errors while changing Boundary Condition name.

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Structural Detailing

TR259540
Miter end cuts for the knuckle profile are not proper at the knuckle connection (part of profile connected at the assembly with overlap/gap) if the profile has a seam point on it.

TR259764
The automatic placement of Generic Member Assembly Connections when detailing Advanced Plate Systems on three standard members in a 'T' configuration results in an error for features between two of the members that have a frame connection of Axis-End to each other.

TR259765
When detailing Advanced Plate Systems on two standard members in a 'T' configuration, with one of the two members being split by the other, results in errors for the automatic placement of Generic Member Assembly Connections.

TR261198
Some 'rathole' parameter rules assume the bounded and bounding flanges are parallel by setting the angle parameter to a static value of 180.
Workaround: The user can manually measure and set the angle of the bounded member.

TR263172
When a Free Edge Treatment (FET) is created using a single boundary, and a solution is selected through ambiguity, then the length of the Edge Treatment in the properties page is shown as zero.

TR263724
Users cannot delete assembly connections when placed on a read-only member.

TR266339
Member end cuts do not reflect change in diameter of bounding custom built-up tube.

TR266870
A physical connection is not generated for a bracket when bounded with a leaf profile (where adjacent leaf profile is not detailed).
Workaround: In order to generate the physical connection, both leaf profiles should be detailed.

TR267448
A chamfer fails to generate correctly between a Built-Up Can and Built-Up Tube after detailing.
Workaround: The user can place a manual chamfer and modify its parameters.

TR268113
While creating lapped plates, if the user selects an edge reinforcement with either a bend or split knuckle as a boundary, then a To Do List entry as “Error when computing this object” is generated.

TR268264
While creating lapped plates on curved shell plates using lapped plates, an error message is displayed and a To Do List entry “Error while computing this object” is generated.

TR268454
End cuts for tubes fail when bounded geometry is similar in shape to a circular arc.

TR269106
When a lapped plate boundary is defined by a sketched contour, the lapped plate part is trimmed such that the edge surface is orthogonal to the sketch plane rather than orthogonal to the plate on which it is lapped.

TR269396
The handrail toe plate of a stair does not trim correctly to post, and a To Do List message is generated stating “Error during execution the Member Construct".

TR270238
After inversion of Logical Connection, existing Assembly and Physical Connections could be removed.

TR271281
When splitting an Advanced Plate System transition plate and then detailing it, the member part is improperly trimmed.

TR273665
When a user places a transition plate on a built-up member, detailing will result in errors.

TR273675
When a user places an insert plate of type CrossInsertPlate1 on a built-up member, detailing will result in errors.

TR275524
An end cut is not generated when the load point of the edge reinforcement is changed.

TR275700
When an opening is placed that intersects a chamfer, the chamfer shape becomes incorrect.

TR276311
Miter end cuts are not proper for profiles. Currently, miter end cuts are created with reference to the molded line.

TR281206
Edge Reinforcement Extend Landing Curve gets reversed when the knuckle property is changed from Ignore to Bend.

TR281849
The user is not able to plan ‘MbrAxis_ToOnTubeMember’ assembly connection for web penetrated cases and ‘MbrAxis_ToFaceAndOutSideNoEdge’ assembly connection for flange penetration cases.

TR283347
Sketched features are failing on Copy/Mirror and Copy by Family commands if the plate is split by design seam. If the plate is mirrored the sketched features on the source plate 'disappears'.

TR283975
If a profile is split and one leaf system is deactivated, the free end cuts are not being created.

TR284860
On the Parameters tab of the Physical Connection property page, the Tail Notes field throws an error if anything is keyed in.

TR288026
While placing an edge feature on a plate/profile using the Placing Edge Features by Offset command, then selecting relative check box in the Offset Table for Edge Feature dialog box, then the values calculated for placing edge features are calculated incorrectly for the third feature and above in the sequence.

TR288399
When changing the corner feature rule on a tripping stiffener, the collar plate goes on the To-Do List.

TR288865
When placing a Generic Member Assembly Connection on a short standard member with Advanced Plate Systems, it trims away the wrong end of the member.

TR288907
For two profile parts that are split from a profile system, when GetOrientation is called on the points on the Extended landing curve that lie on the portions beyond the Profile part landing curve, the origin returned for those points are all the end points of the part landing curve. This affects manufacturing output.

TR289637
When trying to detail a three-support bracket with hull, hull profile, and knuckle plate as supports, there is an error with the trim.

TR289730
User creates deck as linear extruded plate with some plates with profile passing through it. When user details the deck plate cutout is created but corner feature is not created.

TR290528
Unable to trim slanted linear extruded plate that is bounded to a profile web and the profile web is near the flange section at an angle between 0-70 degrees from the profile axis in an anti-clockwise direction.

TR290612
When a user extends the part end by invert option in Logical Connections, part shape is not correct with To Do List- “could not apply final trim to the geometry” and To Do List – “Could not get trim candidate” with the intersecting deck plate.

TR291457
If user toggles between split methods for profile at knuckle incorrect results can be seen.

TR291604
When profile is placed such that it extends beyond the thickness of the bounding plate, setting Offset Method to ‘Offset and Trim with an Offset’ and then detailing the profile, results in trimming on actual profile part rather than the extra part extended beyond the plate.

TR292467
When a plate is unfolded by detailed neutral surface, a warning is seen in the To Do List, “Failed to get data from requested surface. Alternate surface is used for unfolding”.

TR293550
The Neutral Surface function fails to return a neutral surface for unfolding, therefore unfolding does not occur at neutral surface when trying to manufacture the plate part.

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Structural Manufacturing

TR249864
When a knuckled profile with "Ignore" knuckle is manufactured, a knuckle mark is not being created after manufacturing it.
Workaround: User can place a manual marking.

TR263723
The Hawse Pipe manufacturing data is incorrect. The XML output is incorrect for the revolved plate system.

TR265461
Profile length is reported incorrect in Part Monitor output. Profile length is expected to equal the detailing geometry length and the margins, with the Root gap subtracted, but in this specific case the profile length is incorrect.

TR271990
When a knuckled edge reinforcement (with "Knuckle" type ="Ignore") is manufactured, knuckle marks are not shown in the Part Monitor or Plate Cutting drawings. The edge reinforcement is on the To Do List with a warning from "Knuckle" line marking rule, and the knuckle mark is not generated.

TR272197
The face marking of the web for a built-up L profile is not generated on the flange in the Part Monitor output.

TR273251
Openings created by sketched features in the Structural Detailing task are not seen on the pin jig set up.

TR273380
When selecting only a flange (from BUT profile) for a regional margin, the web is receiving it as well.

TR274100
Bevel properties are missing in the Part Monitor output for edge reinforcements having a bend knuckle. Bevel properties can be observed when the knuckle is set to Ignore Knuckle.

TR277335
Incorrect geometry is generated in Nesting machines when exporting a plate part having annotations near its contour.

TR281045
A sketched feature was placed on the web of a member. The member was manufactured with the ProfileUpside value of WebLeft for DSTV, and the output viewed within the Part Monitor showed a section of the flange was missing.

TR281179
A member was manufactured with the ProfileUpside process of WebLeft for DSTV. A bevel annotation was seen to be incorrect when viewing within the Part Monitor. Further investigation within the output xml file found that many of the CVG_POINTs for SMS_BEVEL were identical.

TR283051
The unfolded shape of knuckled bulkhead plate part should have a split outer contour at the knuckled portion of the plate part.

TR283228
Plate and collar location markings are not present on a profile in the Part Monitor output.

TR285914
Manufacturing Panel has trouble with the output creation for Advance Plate System plates containing knuckles.

TR287255
When a user splits a member and applies an assembly connection at the split point, physical connections (PC) are generated. The PC properties have the bevel information but in manufacturing the member, the manufacturing output bevel is missing with the bevel annotation.

TR287272
Incorrect part monitor output of knuckled profile parts in select cases.

TR290832
Currently in order to view new annotations generated by marks placed or modified in the part editor it is necessary to exit and re-enter the part monitor. It should be possible to include a refresh option that will generate these annotations without having to leave the part monitor.

TR290833
When modifying the end cut shape for both the top flange and the web of a profile persistence fails in the part editor due to an issue with closed contours.

TR290966
The Part Editor ribbon bar needs to be updated to include missing information, which cause the creation/editing of contour/marking/annotation to fail.

TR292134
A curved profile was manufactured, and the output was viewed within the Part Monitor. Bending lines were placed on the profile. However, there were bending lines crossing the minimum vertical limit of the outer contour.

TR293578
If the user has multiple coordinate systems in their model, and some of them are at an angle to the global coordinate system, templates placed on these coordinate systems have an incorrect orientation.

TR294550
For Pin Jigs, the pins and remarking lines are missing when generated for appendage plate parts. There are different behaviors seen with different 'Pin Jig Remarking Rule set'.
Workaround: To use 'Based on Logical Connection' Ruleset with Plate Remarking Lines set to Ignore.

TR295215
On manufacturing plate output, no Tangent lines are not shown in manufacturing xml even though the "PINavalArchLines" arc line are set to apply.
Workaround: Manually create markings.

TR295467
When the user changes the Shrinkage working status, the same is reflected on the sibling Manufacturing entities even though they are not modified. When user creates a new Manufacturing entities with working status, other Manufacturing entities working status is changed back to working.

TR295912
User has a reference curve on the Hull part. With "PINavalArchLines" arc line set to apply in manufacturing properties, user is not seeing the curve in the manufacturing part monitor output.
Workaround: Manual marking with 2D projection option reference mark can be seen in the manufacturing output.

TR295929
When user sets the manufacturing method of knuckle on the Edge Reinforcement to Ignore, the knuckle location marks are not in the correct location in part monitor. The overall length of profile is correct.

TR296370
Manufacturing Template contour does not match with the template drawing in the model. There are variations of 2 to 4.5 mm in some templates near the anchor spool section.

TR297048
The outer contour of manufacturing part in part monitor is wavy of bow plate when using TwoDirUnfold algorithm. The manufacturing output is correct and far better when SNU algorithm is used. Currently the complete roll information in both the directions with any of the unfolding algorithms.
Workaround: Use the SNU algorithm (which is default) and rely on template data for accurate bending of plate.

TR297175
The manufacturing part XML is not generated for a specific fore region hull plate part wherein the plate part is in To Do List. The problem occurs during the unfolding while using Detailing Neutral Surface.

TR297773
In model, when the bracket is crossing the plate having chamfer, plate (bracket) location mark does not appear in part monitor output on one plate having lesser thickness. When a 2S bracket is placed, the issue appears on both plates.
Workaround: Place manufacturing mark manually.

TR299009
The manufacturing part XML is not getting generated for flat bars if the symmetry is set to port starboard and symmetry part is left blank.
Workaround: Manually change the part symmetry to Port or Starboard or not specified.

TR302803
If after selecting a plate for the Copy Structure command and then the mouse crosses over a standalone plate part either graphically or in WSE, error message(s) appear. This workflow may also corrupt and/or crash the session.

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Structure

TR125206
Trimming against curved surfaces/members does not work correctly.

TR125736
An opening or any object placed on a wall gets relocated after splitting the wall.

TR127967
Adding/removing a wall run segment moves openings from their initial face.

TR155975
In some cases, the horizontal offset is wrong after the user does a mirror-copy of Ladders or Stairs with a Slab as the top edge support.
TR166214
Slabs cannot be placed using curved members that are pipe or tube cross sections as boundaries.

TR182278
Footings have a compute order problem. The slab does not get updated on pier changes.

TR184346
Slabs are not bounded correctly by curved members with a sweep greater than 180 degrees.

TR188246
The center of gravity for Footings is not calculated correctly after a Model Data Reuse move or transform.

TR197227
Equipment Foundation Slab/Block total length is not correctly calculated.

TR200710
Merged Pier Combined footing edge clearances are incorrect on a rotated Coordinate System.

TR216080
3D sketched slabs go to the To Do List when the elevation plane support location is modified.

TR218744
2L_FP Cross Section does not respect Insulation Spec Thickness.

TR240717
Slabs cannot be bounded with 2D or 3D circular walls.
Workaround: Create circular slabs that are not related to the walls using radial grids or the 2D sketcher.

TR259076
Slots are not being calculated properly due to the incorrect re-symbolization of base plates for plate penetrating plate cases.
The issue is seen in the following cases:

1. When the penetrated plate is inclined to the plate it is bounded to.
2. When the bounding plate is split close to the location where the penetrated plate is bounded to.

TR259764
The automatic placement of Generic Member Assembly Connections when detailing Advanced Plate Systems on three standard members in a 'T' configuration results in an error for features between two of the members that have a frame connection of Axis-End to each other.

TR259765
When detailing Advanced Plate Systems on two standard members in a 'T' configuration, with one of the two members being split by the other, results in errors for the automatic placement of Generic Member Assembly Connections.

TR261198
Some 'rathole' parameter rules assume the bounded and bounding flanges are parallel by setting the angle parameter to a static value of 180.
Workaround: The user can manually measure and set the angle of the bounded member.

TR263205
Brackets fail to place or have incorrect orientation when the edge reinforcement is not centered.
Workaround: The user can manually adjust the bracket tripping rule because it currently assumes the edge reinforcement is centered.

TR267448
A chamfer fails to generate correctly between a Built-Up Can and Built-Up Tube after detailing.
Workaround: The user can place a manual chamfer and modify its parameters.

TR271556
User unable to place an assembly connection for a post (HR cross section) on a curved standard member (I-beam cross section).

TR271666
Standard assembly connections do not have proper clearance when not orthogonal.
Workaround: The user can manually adjust web cut parameters to get the required clearance.

TR273665
When user places a transition plate on a built-up member, detailing will result in errors.

TR273675
When user places an insert plate of type CrossInsertPlate1 on a built-up member, detailing will result in errors.

TR274569
When the assembly connection selection is modified to “Insert member” for web penetrated cases, the member will fail to trim correctly.

TR280046
Incorrect or inconsistent rotation values are shown in the ribbon bar for columns.

TR281849
The user is not able to place ‘MbrAxis_ToOnTubeMember’ assembly connection for web penetrated cases and ‘MbrAxis_ToFaceAndOutSideNoEdge’ assembly connection for flange penetration cases.

TR282551
The GCLineFromCs shows twice the value entered in the length field when the option is "centered".

TR287117
Basic Geometric Constructions should not accept First Class Business Object (FCBO) as input to avoid referencing the latest geometry of the FCBO, causing potential loops. Users and content developers should always use the FacePortExtractor or EdgePortExtractor on a stable geometry instead.

TR287278
Copy by Family symmetry modify will not work properly for a Corrugated Bulkhead which is created using Coordinate System (CS) created by Geometric Construction and Plane created by Geometric Construction.

TR287424
Built-up tubes create a To Do List record when one of their frame connections is a surface default connection to a linear extruded plate.

TR288399
When the changing the corner feature rule on a tripping stiffener, the collar plate goes on the To-Do List.

TR288865
When placing a Generic Member Assembly Connection on a short standard member with Advanced Plate Systems, it trims away the wrong end of the member.

TR288926
The Best Practice Ladder side-frame is not displaying in drawings.

TR289730
User creates deck as linear extruded plate with some plates with profile passing through it. When user details the deck plate cutout is created but corner feature is not created.

TR289940
After performing a Model Data Transform (MDT) of a slab, its boundaries cannot be changed and a To-Do List record is created stating “Defined contour is open.”

TR290482
Error messages result from splitting a standard member that has a detailed profile bounded to it.
Workaround: Undetail profiles bounded to the standard member. Then use the Place Splits command.

TR290858
After a user deletes an opening in Sketch 2D while modifying an opening, Sketch 2D will no longer re-open to allow the user to place a new opening.

TR293211
Inconsistencies are seen in Mirror Copy of designed members and standard members originating from frame connections with the Reflect user attribute. This has an impact on the Copy by Family Create and Copy by Family Modify commands.
Workaround: The user can manually modify the attribute at the target. However, the user still will not be able to use Copy by Family Modify since this command will pick the modified objects as candidates.

TR293253
Slab assembly connections are missing when defining the workspace by volume.

TR293254
When using a topological point (Geometric Construction point at minimum distance) to drive a sketched boundary for a planar plate, the point does not get updated correctly if the plate's plane is changed.

TR293699
Plate Normal direction is changed when user rotates the Grid from N *x.xx* deg E to E *x.xx* deg N causing all the molded conventions to be changed.

TR293771
Cannot place designed member between topological points.

TR294972
When user creates a stiffener on a plate with the Associative offset in 2D Sketch, then when user modifies the plate height, the geometry of element of plate updates but the stiffener is not updated. User has to open the 2D sketch for the stiffener to update.

TR295631
When 4 stiffeners of C_SS cross section are mutually bounded around the Chute, Box cuts are not getting created on short side upon detailing. Box cuts should be placed with proper Trimming without any errors.

TR296444
Generic Assembly connection is not deleted after modifying the frame connection.
Workaround - User has to delete the Generic member AC manually first and then move the frame connection.

TR297832
Split command fails for a member bounding an opening.

TR298158
When the user tries to move the eccentric footings (not any specific footing), the error “Method of Object failed” is thrown and the user will not be able to move the footings. Henceforth, the move command is also disabled till the user opens a new session file.

TR298365

“2S_Rib” bracket shape from Fast Track catalog fails to be generated with SOM.

TR300118
Place Wall Command errors out inconsistently.

TR300164
Access denied errors on trying to delete slabs in GWC.

TR300853
In a specific case, Generic member AC is not created correctly when the plate port selected is not intersecting with either member Web left(WL) or Web right(WR). The web cut created is not trimmed with the bounding Plate port.

TR301468
When creating Advance Plate System with Geometric Construction LineAtAngle, it was found user cannot modify LineAtAngle sketching plane. Currently LineAtAngle does not use sketching plane (third input) if the reference point is not exactly on the reference line. This is causing the Advanced Plate System creation to fail.

TR301701
GussetPlate9 (GCGussetPlate9) will not evaluate when the secondary member (input 3) is of certain cross sections. For instance, when the secondary member is a Built Up I cross section, the Advanced Plate System will fail.

TR302263
In the case of a Geometric Construction plate, where profile mounting angle is not normal to plate, the miter cut creates a gap.

TR302274
Slabs not getting selected correctly by composition.

TR302416
When Standard Tubular Member(Pipe) is connected end to end with Built-Up Cone:

- After placing the Generic Assembly Connection between the two, Default web cut item created is "SnippedTube_Gusset" which is failing to give the correct result. By default, “FlushBoundedDepth" needs to be created as it is correct output.
- The default Physical connection created is "Tee Weld". In this case "Butt Weld" needs to be default.

TR302743
Sketched opening constraint on plate is lost when adding offset to the constraining edge of the plate. The lost constraint impacts Split Root System.

TR302814
In specific model, slot cannot be created at some locations. Sketched curves have been created around the web plate + Edge Reinforcement in order to split the bulkhead. The resulting seam, Logical Connection, and Assembly Connection are created as expected. The slot root selection is failing with error in log.

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Systems and Specifications

TR292934
Only the last Excel spreadsheet is fully processed when multiple spreadsheets are selected in the Import Systems Hierarchy command. All others will appear in the import log file as only read files.

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Translators

TR250335
A specification change is not honored by Smart 3D to PDS export process at components/instruments/specialties.

TR274371
SmartPlant Interop Publisher (SPIOP) currently does not publish Smart 3D fireproofing data to its corresponding member part using the delivered mapping. Instead, it publishes the fireproofing data and its corresponding steel column as separate member parts.

TR293884
Connections are not formed between some pipelines for imported PDS sample data.

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