ThinkDesign Release Notes from v. 2015 to v. 2018

In general, for each release, the source code has been accurately revised so that the program has highly improved its robustness and its reliability has been made much higher.

ThinkDesign 2015 Release Notes

Document Explorer: Improvements of the Model History User Interface

The old fixed model history UI area has been replaced by a new much more flexible GUI object window, which enables users to adapt it to her/his needs.
The new window is called **Document Explorer**. As for the previous versions of ThinkDesign, it will be integrated with the ThinkDesign document, using the same elements, such as *Model Structure* and *Visual Bookmarks*, now called **Tabs**.
The following actions can be performed by the user to arrange and customize the entire structure of the windows:

- Keep the **Document Explorer** window docked, as since the previous versions of ThinkDesign.
- Float both the entire **Document Explorer** window and each single tab pages over the screen.
- Hide both the entire **Document Explorer** and a single tab page.
- Pin the **Document Explorer** window either to the Left or to the Right document border.

- Reset the entire **Document Explorer** window content to the default layout.

Part List Editor Enhancements

Two major enhancements and two minor enhancements have been added to this version.

- The major enhancements consist of providing the **Edit Part Data** command in the right-click context menu and of showing custom properties as Boolean in the part list. These properties will in fact be shown as check-boxes in the part list editor. If the value of the property is 0, then the check box in the Part List for that component will be unchecked, else checked.
- The minor enhancements consist of showing hidden components rows in Italics and restrict data type entry into part list cells while editing.

Mold Standard Part Enhancements and new Multi-Cavity Layout

A new way for creating multiple copies of product/core cavity sets, position and orient them is now available. The new command **Multi-Cavity Layout**, which helps you create a number of instances of product/cavity sets and also to position and orient them as you wish.

New enhancements have been made also to the **Mold Standard Parts** command and to the management of custom parts and threads.

NPT Threaded Hole

The *National Pipe Thread* or *NPT* is an American standard for threads on tapered surface. Till last versions, ThinkDesign performed the feature using cylindrical holes. Starting from this version a specific feature has been developed.

New Mating

The Mating environment has been changed with the goal of providing you with a brand new UI so that the operations of working with constraints is much easier and natural than it was in previous versions. The **Multiple Mating** is no longer available, since the new structure of the mating mechanism is much more flexible and much smarter, so that only

one Mating command will manage the whole kinematic chain, also in the cases previously covered by multiple mating.

Note for the old users
Please take into account that since there is no longer a reference entity and an entity to be positioned, but both entities are involved symmetrically in the application of constraints, do not expect the first entity to be fixed as it was in older versions. To make it fixed, use the Fixed Constraint command.

Old Assemblies and Compatibility
If you load a model containing mating events created with previous versions of ThinkDesign, they will be automatically converted to the new mating structure. So, even if you do not perform any action on the model, when you close it you will be asked whether to save it with the new mating structure. Obviously, if you need to use it with older versions of the package, do not save it. The new mating structure is not supported in older versions, so we recommend making a copy of the file if you need to use it in such versions, or to be very careful not to save it with the current version.

Converters upgrades

The TD XchangeReader module is now capable to read:

- Any SolidWorks© version up to 2015
- Any Autodesk® Inventor® version up to 2015
- Any JT version up to 10.0
- Any Siemens NXTM (Unigraphics) version from V11.0 up to NX 10.0
- Any CATIA® V5 version from R4 to V5-6R2014
- Any CATIA® V6 version from 2011 to 2013
- Any Parasolid® version up to v27.0
- Any Pro/ENGINEER version up to Wildfire 5
- Any Creo™ Parametric version up to 3.0

For the Autodesk® DXF/DWG format, the DWGDirect 3rd party library has been upgraded to Teigha® 4.01.

New 3D Printing command

On Windows 8.1 systems or higher, the 3D Printing command enables you to print to a 3D printer.

New Licensing System

The Licensing System has been completely renewed. See also SafeNet® Sentinel Admin Control Center on the web. In case of need, get in touch with the Customer Care Service.

ThinkDesign 2016 Release Notes

Converters upgrades

Converters have been updated with the following improvements:

- Support of Any Autodesk® Inventor® version up to 2016
- Support of Any CATIA® V5 Version from R4 to V5-6R2015
- Support of Teigha® 4.1.1 for the Autodesk® DXF/DWG format (through upgraded DWGDirect 3rd party library).

ANSYS plug-in: new versions available

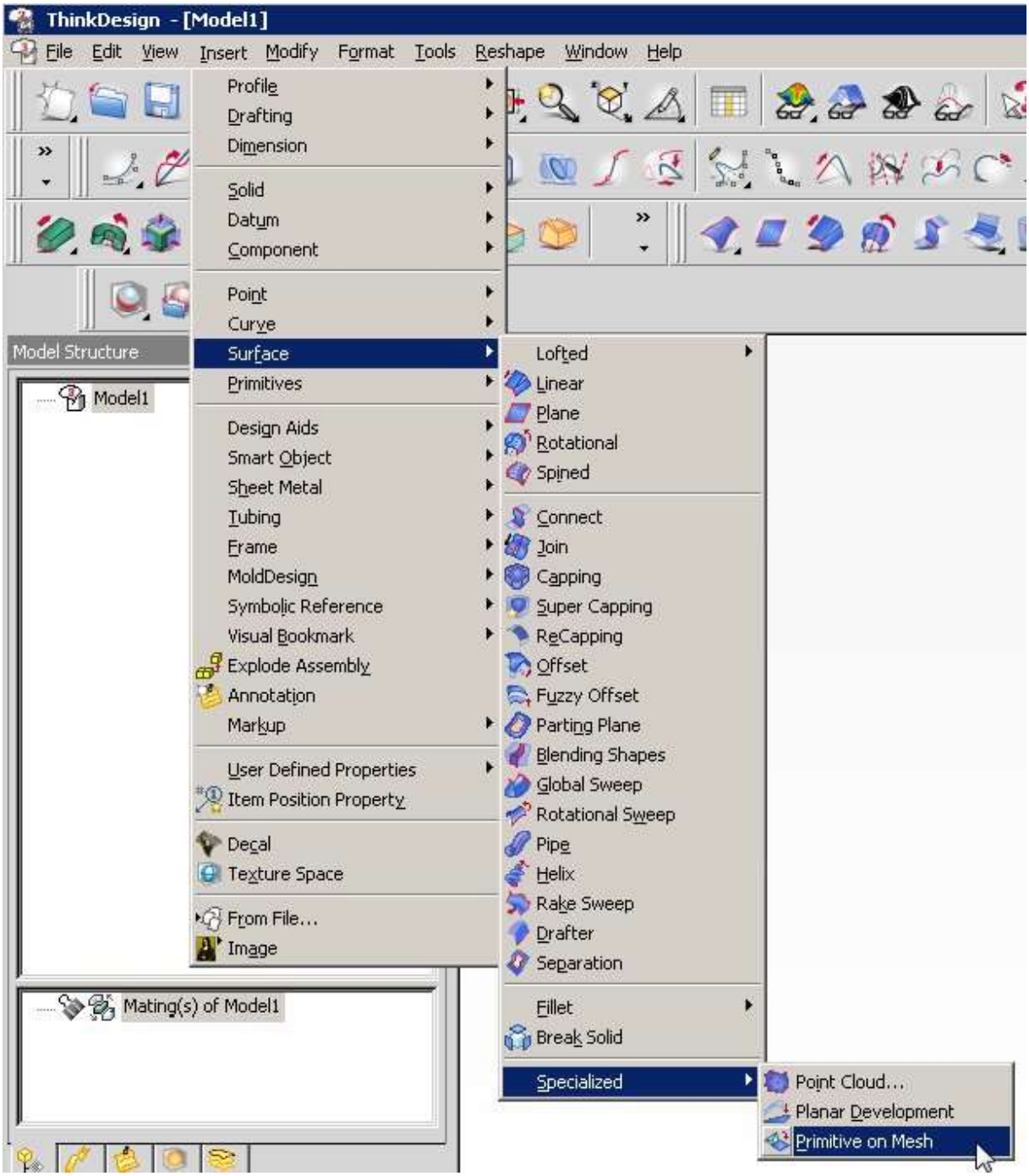
Starting from this version, beyond versions 14.5 and 15.0, you can also choose whether to install either ANSYS V16.2 or ANSYS V17.0 plug-in from the custom window of the installation wizard during the installation process.

Windows® 10 Support

ThinkDesign can now be used also under Windows® 10.

New tools for targeting Full Hybrid Modeling with meshes

The new **Primitive on Mesh** command enables designers and engineers to solve spatial, interactive and creative design tasks in immersive, collaborative environments while maintaining CAD modelling as a familiar design method.



The command enables you to create at the same time several shapes such as planes and cylinders that best fit mesh facets, and to add constraints between the created shapes. In the list of constraints, it is possible to specify, for example, a cylinder must be orthogonal to a plane and the command will provide you with the best solutions based on their tolerance values. For example, you might also have several cylinders and specify they must all be parallel.

Other new tools:

- Smart selection of mesh facets in two different modes:
    - Crease Angle
      The **Crease Angle** command enables you to make mesh facet selection using

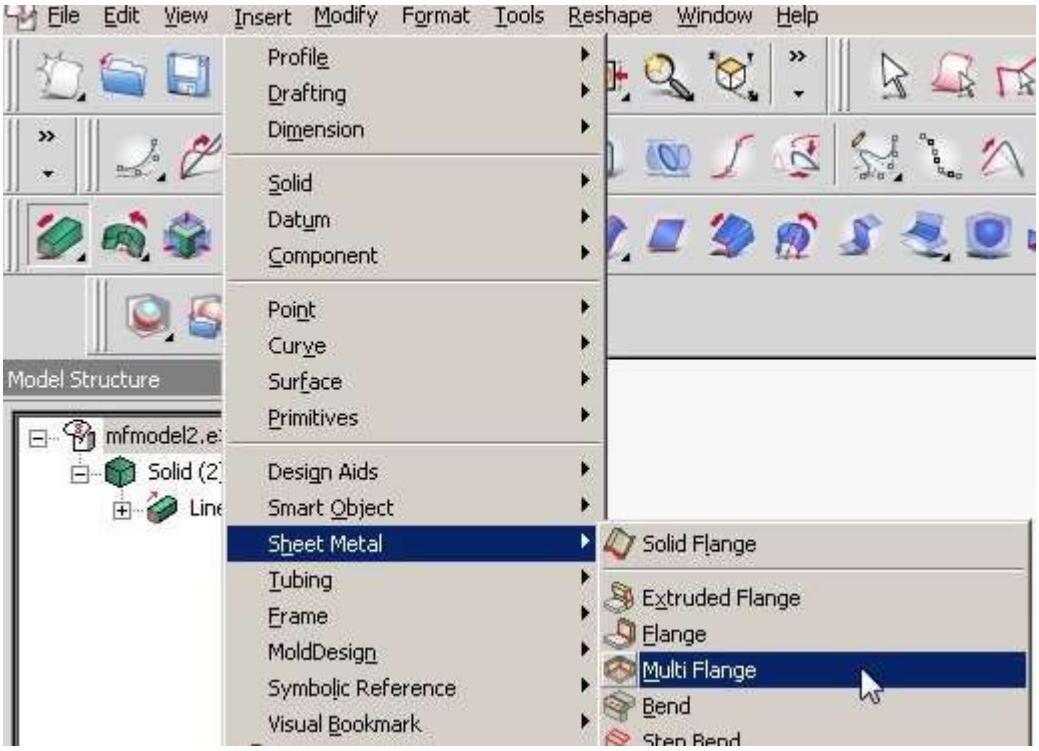
as criteria the angle between the normal of the picked facet and the normal of the other facets of the mesh.

- Geodesic Distance
  The **Geodesic Distance** command enables you to make mesh facet selection using as criteria the distance between the picked point on the facet and the other facets of the mesh.

- The **Select Facets in Front Only** command is used during the selection of facets by Window or Lasso. By default, it is OFF: all the facets in the window or lasso area are taken in the selection. When the option is ON, only the front facets are taken in the selection.
- New **Snap** functionality for the mesh nodes.

New Multi Flange command

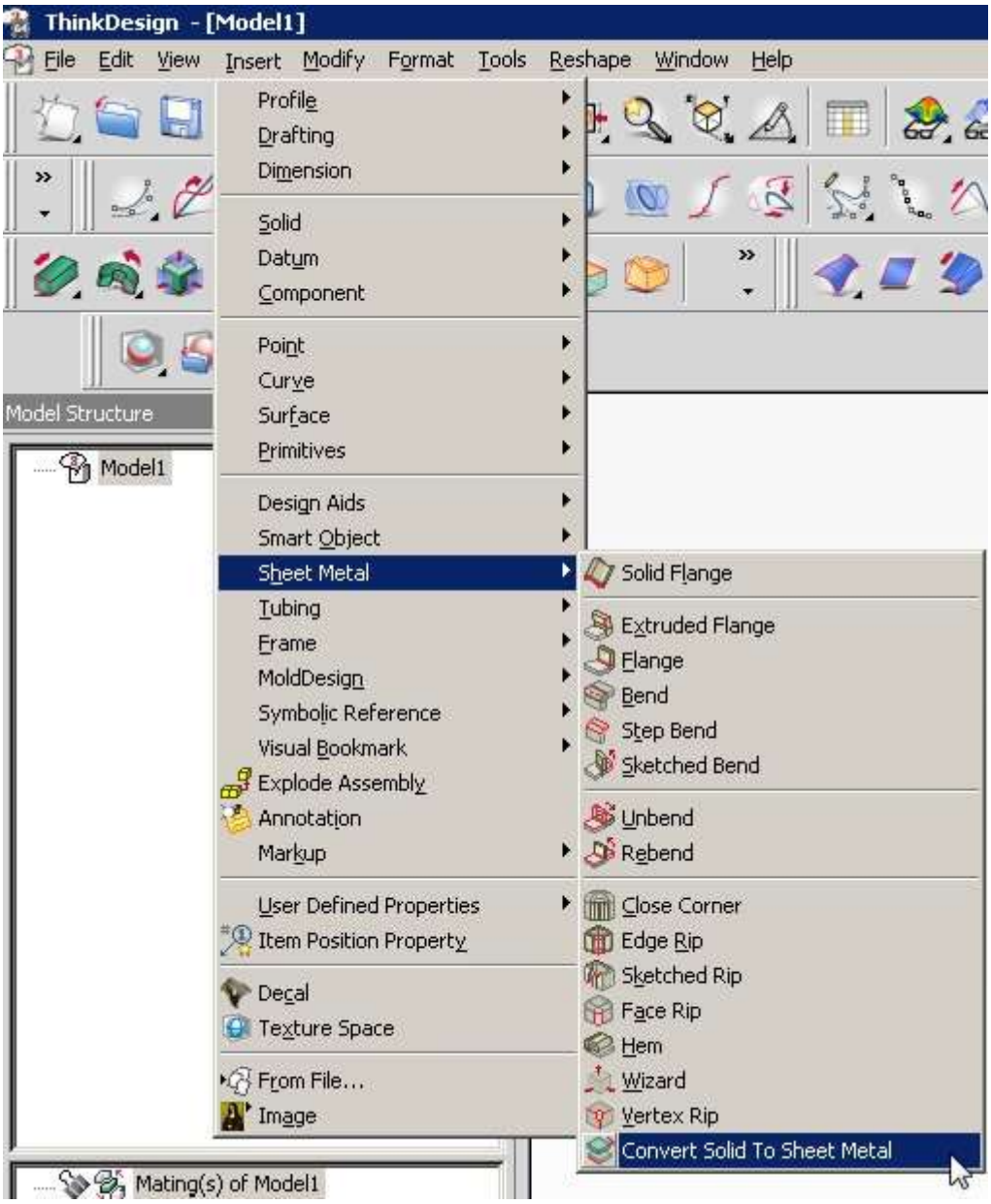
The new **Multi Flange** command enables you to create multiple flanges within the same command session.



At the same time, you can select separate chains with discontinuity points, and they will be correctly managed by the command with the proper miter gaps.

New Convert Solid to Sheet Metal command

The new **Convert Solid to Sheet Metal** command has been developed. Using this command, it will be very simple to transform an existing solid into a Sheet Metal entity, thus being enabled to rapidly and easily obtain the Flat Pattern View of the selected solid.



Mold enhancements

Several enhancements have been done for the Mold environment:

- Redefinition of the **Trim** option in **Insert Standard parts** redefinition
- The **Multi-cavity Layout** command creates a pattern and at the end you have several instances, but in a sense everything can be made "associative" from the very beginning, starting from the shape you want to mold, and all this procedure is based

on derived models. Yet, you may want to proceed with design also beyond the command result. In previous version, you might encounter an issue when the first copy of the pattern was taken into account, but it did not impact the other copies in the pattern.

So, the **First copy derived** option has been added:

- If the option is not selected, new copies will be derived directly from input selection.
- If the option is selected, input selection should be a derived component (other selections are not allowed) and new copies will be derived from the X-Reference of selected component.
  This will help you to create multi cavity layout where input selection should be modified but changes should not need to be propagated to multi cavity layout copies.
  This option cannot be changed using **Redefine**.

- The **Rectangular balanced** placement mode has been added to the command
- Mold base template files upgraded to the new Mating except Pedrotti Euro models
- Mold base works without MS - Office installation also.

Miscellaneous

Other improvements:

- Performance of New Mating has been strongly improved. The command is much more stable and reliable.
- 2D Thread representation of NPT holes have been greatly improved.

Help updated to 2016

ThinkDesign online help is now up-to-date with version 2016.

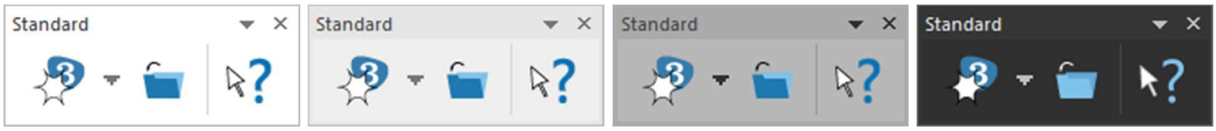
ThinkDesign 2017 Release Notes

New User Interface

ThinkDesign 2017 introduces a new UI:

- **Modern look and feel,** which includes new icons, smart docking style and latest UI controls.

- **Themes** enable the user to apply pre-defined colour themes thru all the set of the UI objects, like Toolbars, Dialog Boxes and so on. Four themes are available ("White", "Light", "Dark" and "Black") and can dynamically be changed directly in **Option/Properties ⇨ System Options ⇨ General** in the Theme choice.



- **High DPI support** provides a consistent look across a different variety of high DPI display settings. All ThinkDesign UI is scaled consistently with the settings chosen in the Windows Display control panel.
  Concerning graphics, the check boxes **Option/Properties ⇨ System Options ⇨ Graphics ⇨ High DPI Scaling for Markers / High DPI Scaling for Lines** enable to scale accordingly also such part of graphics, to follow UI (for example minidialog text) size scaling or not. These settings have some interest with high DPI display that need scaling above 100%.

- **Enhanced File Dialog box** ThinkDesign 2017 offers an enhanced File Dialog box, based on the Windows Explorer UI structure for opening existing file (**File ⇨ Open**) and file saving (**File ⇨ Save As**).
  ThinkDesign 2017 also install the right Windows Explorer Extension to enable the Windows Explorer Preview panel to show .e3 and .2 files preview.

Full Hybrid Modeling

ThinkDesign 2017 introduces the following general enhancements to Full Hybrid Modeling functionalities:

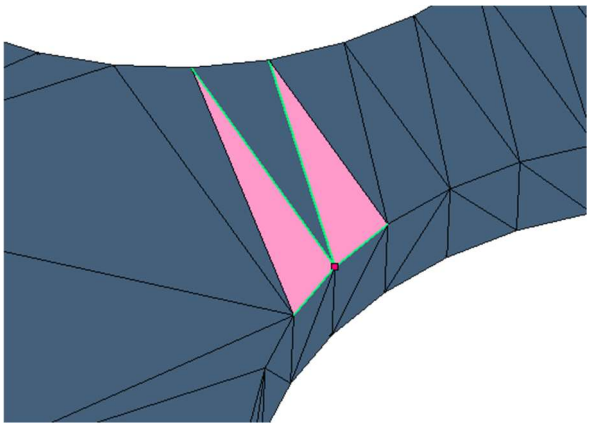
- New "Mesh Repair" command
- New "Remove Facet" command
- Enhancements in View and Zebra commands using meshes
- General improvement of reliability and robustness in the management of poor quality meshes

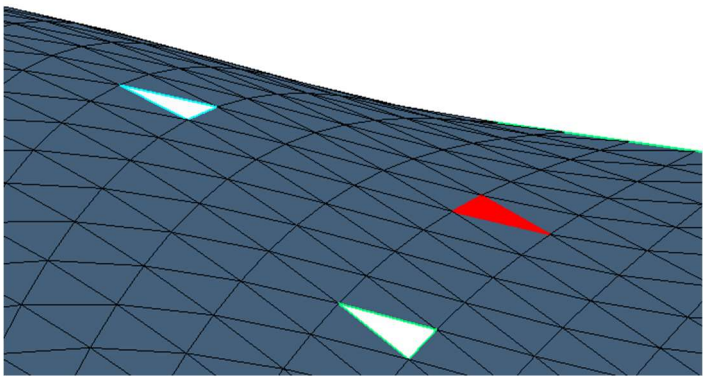
New "Mesh Repair" command (Modify ⇨ Mesh ⇨ Repair)
This command enables the user to check the quality of a given mesh and repair its defects.
With improved quality meshes, commands such as **Compensator**, **Super Capping** or **3D Printing** are more reliable and efficient.

In order to repair a mesh with defects proceed as follows:
Start the Mesh Repair command and select the entity to repair.
A preview is displayed with the boundary edges in green.

Under the **Statistics** node, there is some information about the mesh: number of nodes, number of facets, number of shells, number of boundary edges and the water tightness status. These statistics are updated in real time along the repair process.

Under the **Defects** node, are listed all the defects that can be visualized on the mesh: isolated nodes, duplicated nodes, duplicated facets, degenerated facets, flipped facets, non-manifold nodes, non-manifold edges, shells, holes.





Given a defect type among the list above, the user can:

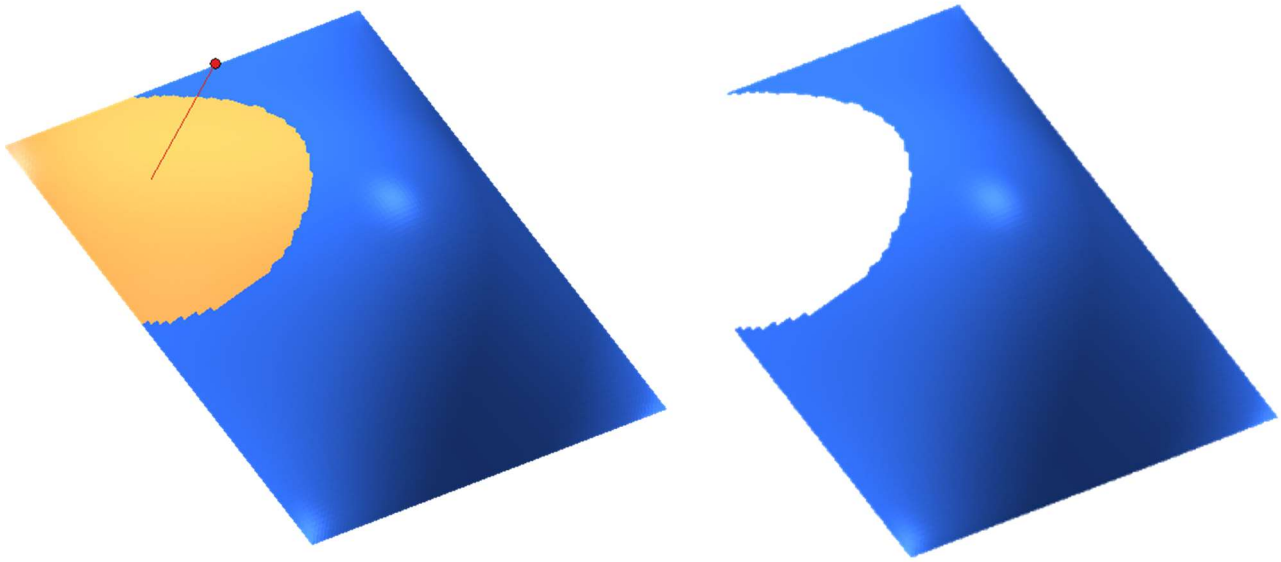
- Change the color of display of these defects using the color button,
- Check the box to visualize and repair these defects: the number of defects is displayed as well as a **Repair all defects** button. If there is no defect to repair, the button is grayed,

- Navigate the defects. Between **Previous** and **Next** well-known buttons, there are 3 other buttons: **Fit view to current defect**, **Isolate current defect** and **Repair current defect**.

The More Options give you additional functionalities:

- **Automatic Zoom** - to adapt the display to the current defect when navigating:
  - When selected, the current defect fits exactly within the display.
  - When cleared, the display is unchanged when navigating the set of defects.
- **Set Target Position** - to set a target position on the current defect when navigating:
  - When selected, the current defect is the target position.
  - When cleared, the target position is unchanged when navigating the set of defects.
- **Isolate** - to isolate the defect from the rest of the mesh when navigating:
  - When selected, only the current defect is visible.
  - When cleared, the current defect is visible as well as the mesh in the selection.

New "Remove Facets" command (Modify ⇨ Mesh ⇨ Remove Facets)
This command enables the user to remove some facets from a triangular mesh. When applying, the facets are removed or become facets of a new mesh if the Split option is checked.
Note that the smart selection commands for facets of mesh (**Crease Angle** and **Geodesic Distance**) are very helpful.



Enhancement in View and Zebra commands using meshes

- **Set Target** and **Set Direction** commands enable now to pick points lying on the facets of a mesh. Respectively, the point on the facet is the new target point of the view and the view direction is aligned with the normal to the facet.
- **Zebra** command supports fully the selection of meshes. The computation and display of the isophote lines has been significantly improved.

Multi-core Management

Multicores processors allow enhanced performance by simultaneous processing of multiple tasks. ThinkDesign can make use of such multiprocessing in algorithms subject to parallelism. ThinkDesign 2017 applies this to GSM Dynamic Previews and Mesh updates (big tessellation), and to GSM Approximation and Convert to NURBS Surfaces (many surfaces).

- GSM Dynamic Previews and Mesh updates
  GSM Dynamic Previews using multicores is automatically activated in GSM family commands: Advanced, Bend, Radial Bend, Twist, Planar and 3DBouding-Box. For "Advanced GSM" command, it means in "Matching interactive" or when using a Mesh as a deformed entity. The update of GSM Dynamic preview is computed in parallel for all nodes of the tessellation/Mesh: the biggest the tessellation preview is the more performances benefit compared to execution in sequence.

- GSM Approximation and Convert Surface to NURBS
  Parallel treatment is automatically applied for surface approximation in GSM family commands and "Convert surface to NURBS" commands. For "Convert surface to NURBS", it is available only on "Advanced Mode". The biggest the number of surfaces is the more performances benefit compared to execution in sequence.

3D profile enhancements

The 3D profile now supports NURBS curves usage in multiple constraints environments:

- Definition of NURBS curve under constraints,
- Dragging NURBS curve under constraints,
- Usage of 3D profiles containing NURBS in commands such as Pipe.

Solid Subdivision Improvement

ThinkDesign 2017 introduces significant performance improvements in the commands Convert to/Modify Subdivision Solid. The new algorithm takes fully benefits of Subdivision theory to make faster the resulting solid. The gain is huge for solids with big number of faces.

Mating Preview Improvement

The Mating command now supports the Dynamic Delay setting. This option enables to show the preview only when the mouse stop moving.
It improves the performances for large assemblies by preventing to compute useless

mating between two intermediate mouse positions.
The dynamic delay choice is available in **Option/Properties ⇨ System Options ⇨ Graphics ⇨ Highlight** check boxes **With Delay**. It can be set ON in Standard mode, or can be set ON in Large assembly mode. In the latter case, the user needs to make sure the Large assembly mode is activated in **Option/Properties ⇨ System Options ⇨ General** check box **Large assembly**.

Converters upgrades

Converters have been updated with the following improvements:

- Support of Any Autodesk® Inventor® version up to 2018
- Support of Any CATIA® V5 version from R4 to V5-6R2017
- Support of Any SolidWorks© version up to 2017
- Support of Any Siemens NX™ (Unigraphics) version from V11.0 up to NX 11.0
- Support of Any Parasolid® version up to v30.0
- Support of Any Creo - Pro/E version from Pro/Engineer 19.0 to Creo 4.0

ThinkDesign 2018 Release Notes

Converters upgrades

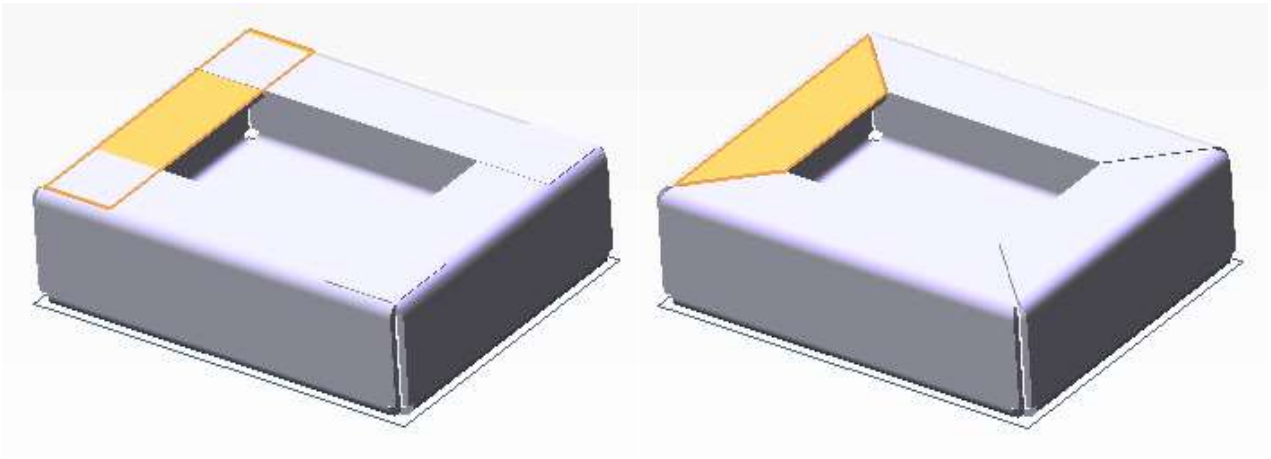
The **TD XchangeReader** module is now capable to read:

- Any **SolidWorks** version up to **2018**

For the **Autodesk DXF/DWG format**, ThinkDesign has been upgraded to be compatible with Autodesk® **DWG™ 2018** file format.
For the **GBG DraftMaker format**, ThinkDesign has been upgraded to import **Raster Image** reference.

Multi Flange Overlap Option

ThinkDesign 2018 introduces a new **Overlap** option in Multi Flange command, available under More Options. With this new option all flanges are created as overlap, while by default flanges are trimmed at the intersection corners.



3D Manufacturing Format (3MF)

The 3D Manufacturing Format (3MF) is a 3D printing format that allows design applications to send full-fidelity 3D models to a mix of other applications, platforms,

services and printers. ThinkDesign 2018 enables now to import and export 3MF files.

New User Interface

ThinkDesign 2018.1.SP1:

- The medium size option for toolbars is available.

ThinkDesign 2018.1.SP3:

- **Themes with Color** enable differentiation of solid features and components by a specific color. This applies to toolbars buttons and Model Structure icons.
  This can dynamically be changed directly in **Option/Properties ⇨ System Options ⇨ General ⇨ Color** check box in the Theme choice.