ThinkDesign Release Notes from v. 2012 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 2012 Release Notes

Switch

The **Switch** tool has been changed. It now allows contemporary management of different versions, old and new ones, of the ThinkDesign Suite products (for instance, versions 2008.x, 2009.x, ... 2012.1). In addition, the new **Switch** tool can manage the product versions both on 32-bit and 64-bit platforms.

Installation and Licensing

Silent installation

A silent installation of the think3's software products from version 2012.1 onwards, can be now performed. This is a typical requirement of organizations, where background processes without any user interaction can be arranged.

Licensing

Starting from version 2012.1, a new licensing algorithm has been introduced. For permanent licenses, the one for version 2011.1 enables all the previous versions, while the

one for version 2012.1 enables only version 2012.1. For temporary licenses, the 3.0 — the version conventionally used to enable subscriptions — enables all versions up to 2011.1, while version 2012.1 enables all versions starting from 2012.1. In order to check how to calculate licenses, please consult the specific area in the think3's Customer Care website.

New optional module TD XchangeReader

The new module TD XchangeReader is available for the installation in the **Custom** page of the installation wizard for both 32 and 64 bit. It allows to read Catia® V5 (*.CATP*) files of any version up to V5 Rel 21 simply using the **Open** command and its **Options**.
The new module is based on **HOOPS Exchange** software by Tech Soft 3D.

Enhancements in Sheet Metal

When viewing a Sheet Metal object in **Flat Pattern View** and selecting **Entity Properties ⇨ Drawing View ⇨ Flat Pattern** from **Options/Properties**, the new option **Supplementary angle** is now available. When selected, the option enables to display the supplementary angle of each bend.

The **Insert ⇨ Sheet Metal ⇨ Step Bend** command shows the number of steps as a driving dimension. This allows the user to edit the number of steps without redefining the feature, for instance by the usual **Edit Driving Dimension** dialog box, entering a different number of steps in the Value box of the dialog box. It also allows the user to assign a variable to the number of steps.

New colored edit boxes now enable users to introduce the variable of the associated entity, close to the box. This occurs in the **Edit Properties** categories **Sheet Metal-Bend Reliefs**, **Sheet Metal-Corner Reliefs**, **Sheet Metal-Flange Corner**, and in the **Reliefs** dialog box of the Sheet Metal commands.

By setting **Bend table** as **Allowance**: in the **Sheet Metal** category of **Entity Properties** and selecting the **From table** option close to **Inner radius**, the corresponding drop-down list shows the available inner radii in the **Bend table** for the **Material/Thickness** combination indicated. The default value displayed is the one of the lowest inner radius. Starting from version 2012.1, you can define the default value best meeting your requirements by the new **Radius** option — displaying a drop-down list — in the **Bend Table Manager** dialog box. By selecting a row of the table of the **Bend Table Manager** dialog box, you can choose the inner radius among those available in the drop-down list displayed for the **Material/Thickness** combination selected. This inner radius will appear as the default **Inner radius** in the **Bend** area of the **Entity Properties-Sheet Metal** dialog box. The drop-down list of the **Radius** option can be viewed only when selecting a row of the table, otherwise it is grayed.

Miscellaneous

New and New From commands

In the **File ⇨ New From** submenu you can find, depending on the context, the **Model from Template**, the **Model Derived from Current**, the **Drawing from Template** and the **Drawing from Current Model** commands to open directly Models and Drawings, and the **Change Current Template**.
The **File ⇨ New**... command now opens the ThinkDesign dialog box for choosing **New File** and **Recent Files**.

New name and position for Recent Files

The **Recent Files** command is now named **Open Recent Files**, and in the drop-down menu displayed by selecting **File** from the menu bar, the command is currently positioned after **Open...**.

New option in the Approximation by Lines-Arcs command

The **Insert ⇨ Curve ⇨ Approximation by Lines-Arcs** command now includes the new **Update deviation** option, under **More options**. When selected, this option enables the new curves being created - that is, lines and arcs - to have the deviation equal to the **Linear Tolerance** value provided. When the new option is not selected, the new curves keep the deviation of the original curves.

Palette saving in DXF/DWG files

The way of storing color palettes in saving DXF/DWG files has been changed. When saving a file in DXF/DWG format, and selecting **Advanced** in the **DXF/DWG Options**, the new **Use AutoCAD Color Index (ACI)** option is now available in the new **Save color palette** area. By default, the new option is not selected, that is entities in the drawing are saved with colors in the RGB standard representation. When the option is selected, entities are saved with colors defined not by the usual three RGB components but by an index representing the AutoCAD internal color (ACI).

New PDF export for models

Upon saving, models as well can be exported to PDF format.
This functionality is based on **HOOPS Publish** software by Tech Soft 3D.

New version of FlexNet licence manager

A new version of FlexNet (formerly FLEXlm) licence manager has been released. It can manage the new licensing algorithm created for 2012.1.

New reader for Parasolid and Pro/ENGINEER files

The **TD XchangeReader** module enables to read also **Pro/ENGINEER** (*.prt*; *.asm*; *.neu*) and **Parasolid** (*.x_*; *.X_*) files, by simply using **Open** command and its **Options**.
For **Pro/ENGINEER**, it enables to read any Wildfire™ version up to 5; for **Parasolid**, any version up to 25.

MoldDesign: enhancements in the standard parts management

The name of the standard parts **Screw** or **Sprue bush** inserted by the **Insert ⇨ MoldDesign ⇨ Standard Parts** command, now follows the rule: "name" and "line number" identifying the object in the catalog — similarly to what is obtained by placing the same object with **thinkparts** — plus a "numeric identifier". In the case of multiple occurrences in the same grouping, the name of these components is the same. This new behavior — shared name — would not allow to modify instances separately. Therefore, the new **Move to New Standard Part Group** command has been added to the context menu on the component, in the **Model Structure**. This command removes the component selected from the grouping and it creates another occurrence with a different name, in order to allow editing.
In the **Standard Parts** command, a new **Insert Component as X-Ref**. check box has been added under **More Options**. When selected, an external reference component is created for each component with a different name, in the same folder of the model. By applying the **Redefine Feature** command, the option appears grayed.

ThinkDesign 2013 Release Notes

Assembly

In the previous version, if you unlinked a simple component, it was renamed using the generic name: *comp<progressive_number>*.
In the 2013.1 version the **Unlink** command renames the component, when the **Enable** check box is checked, following the settings available in the **AutoRename settings** area of the **Assembly/Shared Group** category of the **System Options**, otherwise the new name is *<component_name><progressive_number>*.

ANSYS plug-in

In this version, you can install the **ANSYS V14.5** plug-in from the custom window of the *installation wizard* during the installation process. It is now available on both the 32-bit and 64-bit environments.

Data Converter

In the previous version the TD XchangeReader module allowed to read only **Catia**® V5 (*.CATP*), **Pro/ENGINEER** (*.prt*; *.asm*; *.neu*) and **Parasolid**® (*.x_*; *.X_*) files.

Now it enables also to read **SolidWorks**© (*.sld;*.SLD*), **Autodesk**® **Inventor**® (*.ipt*; *.iam*), **JT** (*.jt) and **Siemens NX**™ (Unigraphics) (*.prt) files, by simply using **Open** command and its **Options**.

For SolidWorks©, it enables to read any version up to 2012; for Autodesk® Inventor® any version up to 2013; for JT any version up to 9.5; for Siemens NX™ Unigraphics), any version from V11.0 up to NX 8.

In addition to the versions supported in ThinkDesign 2012.1, it allows to read Catia® V5 (*.CATP*) files of Rel 22.

Entity Properties

Four new items have been added to the **Type**: drop down list of the **Dimension-Text-Font & Frame** category in the **Entity Properties** tab of the **Options/Properties** dialog box.

When you choose one of them and run the **Label** with the **Multiline Mode** option checked, you get the following results:

JIS Underlining	The underlinings are as long as the longest text and they are joined by a vertical line
JIS Underlining (not longest)	The underlinings are exactly as long as the text above and they are joined by a vertical line
JIS Underlining (not vert.)	The underlinings are as long as the longest text and there is no vertical line
JIS Underlining (not vert.+ not longest)	The underlinings are exactly as long as the text above and there is no vertical line

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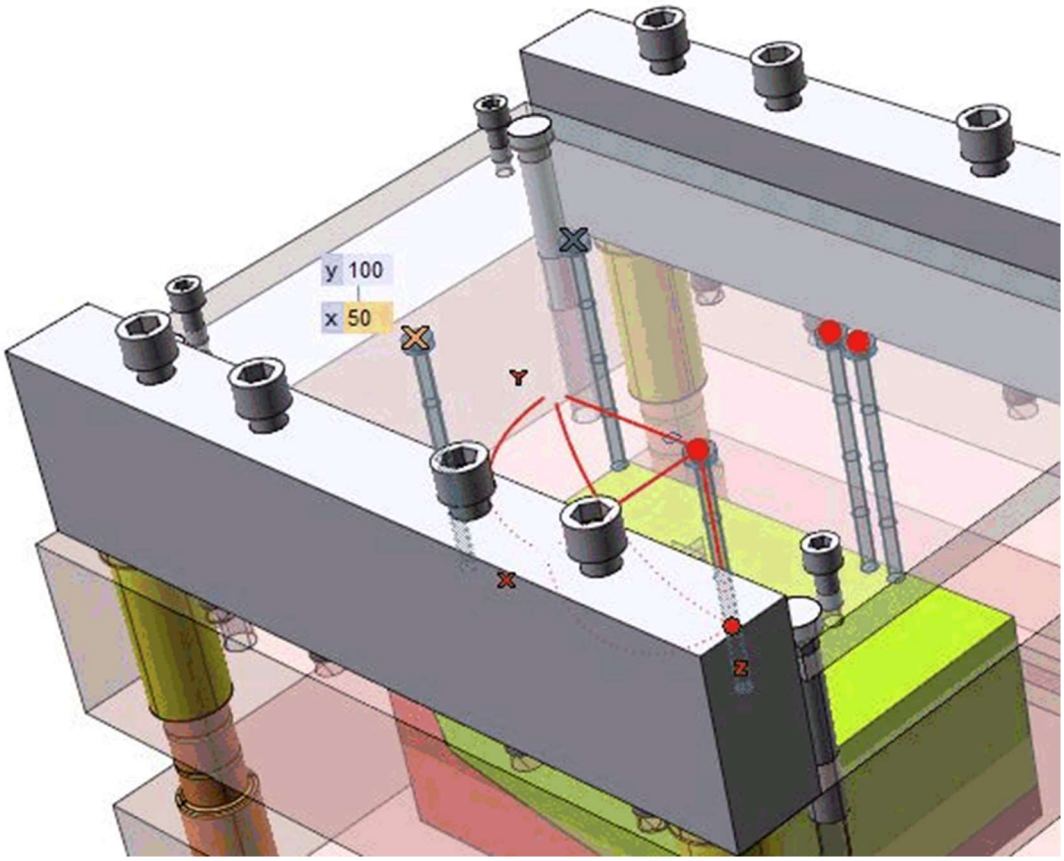
MoldDesign

The **Mold Standard Parts** command now allows selecting as **Positioning Points** not only snapped points but also *free* points on the Work Plane. The graphical feedbacks are different: each snap point is displayed using a red ball, each free point is displayed using a grey cross marker.

This enhancement allows you to position the standard part in any point (*free* positioning) and then, by clicking on its anchor point, to move it specifying the X and Y values or by dragging it. The coordinate mini-dialog boxes are available only if you select one standard part. In the other case (snap) you can change the positioning point to free by double clicking the anchor point or selecting it and use the **Unlink** command in the context menu.



The new **Delete** command is now available in the context menu to remove one or more standard parts. Using SHIFT click you can select more than one standard part; if the multiple selection does not contain any free positioning both the **Unlink** and the **Delete** commands are available in the context menu, otherwise only the **Delete** command is available.

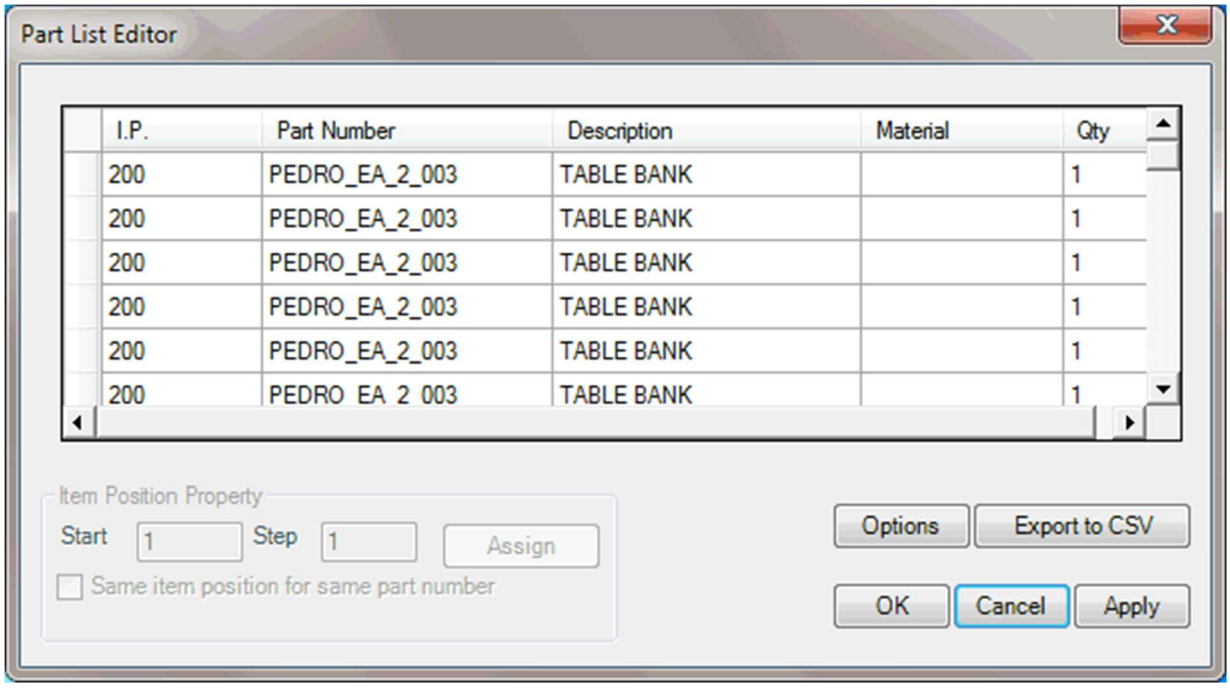
The new **Clearance** drop-down list has been added to the **Cooling** command under **More Options**. You can define how to choose the components in which cooling holes should be created. The options available are:

- **Automatic**
  When selected, cooling holes are created in all the parts that the cooling profile passes through.
- **Manual**
  When selected, the cooling holes are created in only the selected parts. The **Parts** selector is used to select the parts.

Another important improvement is the new **Part List Editor** dialog box which allows you to edit the part list simply.
In the 3D environment, it displays a preview of the part list (without grouping).
You can sort the table as you prefer.
It also enables you to drive the part sequence numbering of the components.

Part List and Ballooning

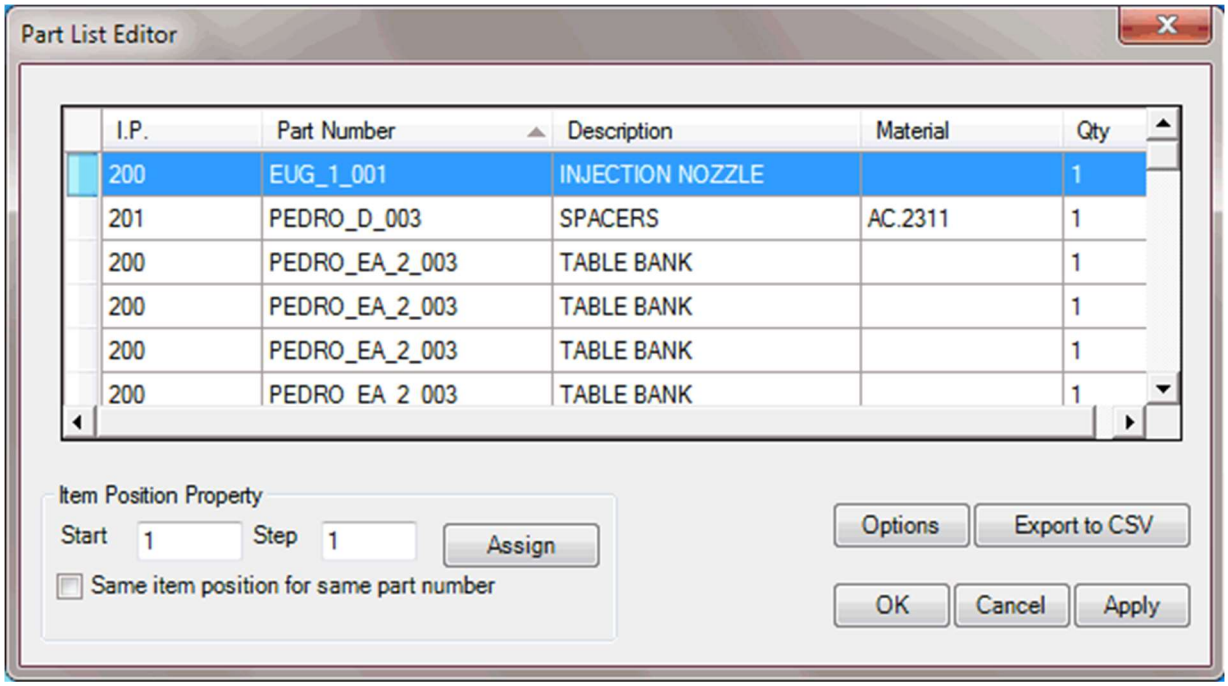
In the previous version if you had an assembly with a hierarchy structure, you could use the commands for numbering, associating balloons and creating part list only for the top level. Now several improvements in these commands allow working on the leaf level too.
The **Hierarchy level** drop-down list has been added to the **Insert ⇨ Item Position Property** command; you can choose between **Top** and **Leaf** and the following selection will take into account your choice. When the **Top** level is selected, the renumbering is done only for the components selected. Similarly, when the **Leaf** level is selected, the renumbering is done only for the leaf level components of the selected components.
In the model environment, the command now allows also multiple selections when **Sorting** is set to **By part number** or **By insertion order**: the new **Selected Components**: selector is available.
The **Hierarchy level** drop-down list has been added also to the **Automatic Balloon** and **Manual Balloon** commands.
The **Leaf** value has been added also in the **Hierarchy level**: drop-down list of the **Part List** category in the **Part List Options** dialog box. In the same category you can choose the character you want to use as **Separator**: when you export the part list to a file.
Also the **Tools ⇨ Part List** command contains the same enhancement in the **Hierarchy level**: drop-down list.
In the model environment the **Tools ⇨ Part List** command now displays the **Part List Editor** dialog box instead of the Selection List.



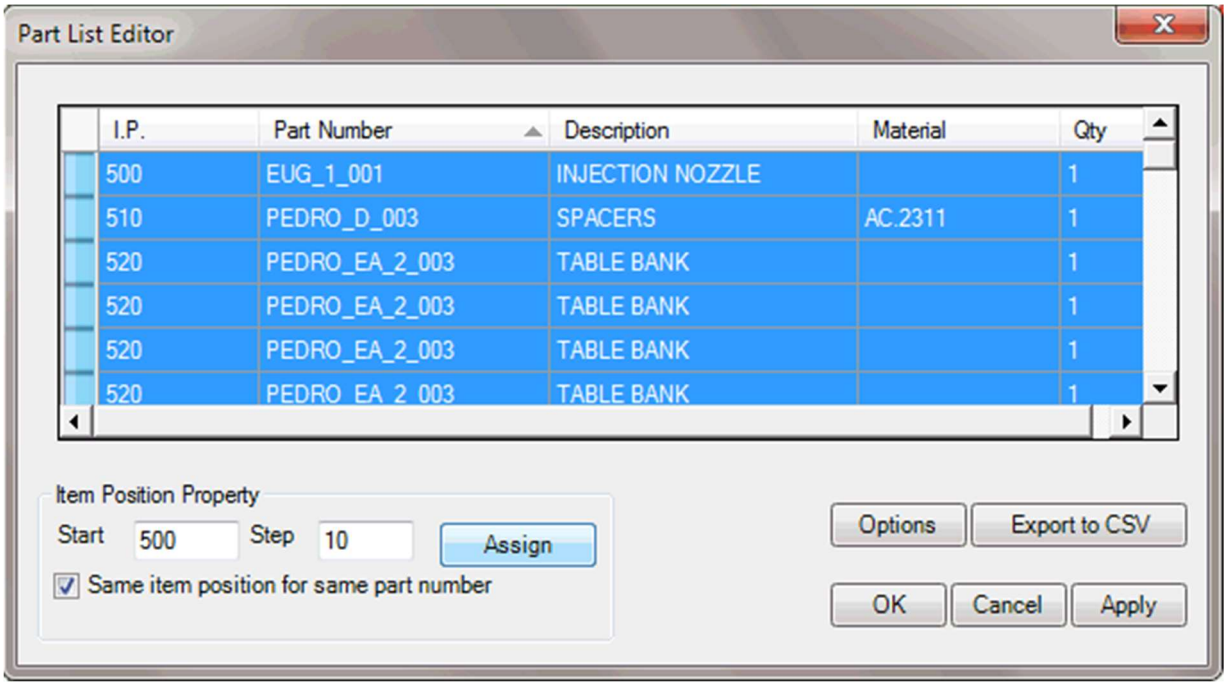
It contains the list of each component and its properties without grouping, differently from the usual part list.
The **Export to CSV** button allows to save the part list to a CVS file, you can choose the path through the system standard file selection box. The **Options** button displays the **Part**

List Options dialog box that enables you to review and change the properties of the Part List.

If you are a MoldDesign user, you can also edit the part list.
The table can be sorted based on ascending or descending order of a column data just by clicking on the relevant column header.
By clicking on the part number of one component you can edit it.
After selecting a row by clicking on it, the Item Position Property area is enabled. You can enter the number you wish to assign to the first component of the selection (Start) and the numeric difference between two successive components (Step) and you can define the behavior in case of multiple occurrences of the same part number (Same item position for same part number).



You can perform multiple selections. If the rows are contiguous, by holding the Shift button and selecting the start and end row. Otherwise you have to hold the Ctrl button and you have to select the rows. To select all the components, you just click on the top left corner of the table.
The part list is updated after pressing the Assign button.



Another important feature is the Zoom Entities command in the context menu; after selecting a component, it increases or decreases the zoom magnification so that selected entities fit exactly within the current display.
By pressing the Apply button you can save the changes without closing the dialog box; with the OK button you save the operations and close the dialog box; by pressing the Cancel button you discard all the operations and close the dialog box.

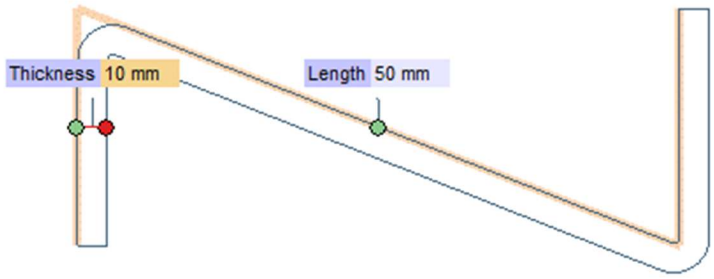
Performance Improvements

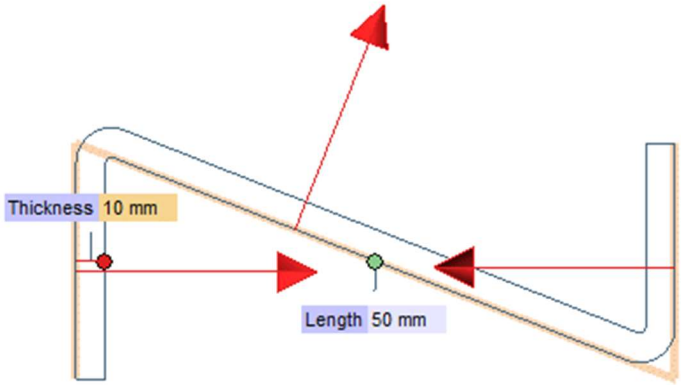
In this version several improvements have been made to increase the performance of ThinkDesign in different and complex situations.
Move with Copies selected on surfaces and faces is faster. For every entity, the higher the number of copies and the complexity of the entity, the higher the gain is. Creating 20 copies of a solid with more than one thousand of faces is around twice as fast.
When rebuilding very complex solids (thousands of faces, hundreds of features) with Propagate properties option checked, computation time is reduced (less 70% on average).
When you create many section curves on a complex solid using the Color increment and Layer increment options, you get the result faster.
When modifying a subdivision solid through moving faces, edges or vertices, reduced the redraw time (by up to 97%), this is especially visible whilst handling big models with thousands of points on the grid. Also extruding a face takes much less time (by up to 93%), this improvement is noticeable on smaller models with hundreds of points but is more flagrant on big models.
When you create a drawing from a model which contains many components (approximately more than 100) and/or solids with many features (approximately more

than 100), you get it faster the first time and every time you rebuild it.
These improvements are visible in both the 32-bit and 64-bit environments.

Sheet Metal

In the **Solid Flange** command is now possible to hold in check the thickness direction of each edge. Because of this you can create your profile and make it parametric without considering the thickness on the single edge.
The new **Local Thickness Direction** check box has been added to the **Solid Flange** command. When checked, the thickness direction of each edge is displayed as a red arrow. You can change the thickness direction of an edge by double clicking on its arrow.





The **Local Thickness Direction** option is allowed only for straight line profiles; in the other cases you get a warning message.

Solid Features

In the previous version of ThinkDesign, when after selecting a **Simple** hole created with the **For screw** check box cleared you run the **Thread** command, after the completion of the command you add the **Thread** feature to the hole. The **Model Structure** changes as you can see in the following image.

In the v. 2013.1 instead, in the same situation, the program executes the **Redefine Feature** command on the hole and automatically checks the **For screw** option e selects **Threaded** in the **Hole type**: drop-down list. This behavior allows to assure the creation of standard threaded holes.

In case of **Simple** hole created with the **For screw** check box cleared and **Blind** as **Extension** and the **End angle** option checked, the behavior of the program is the same of the previous version.
In order to allow the same kind of threads available in the **Thread** command, the **Mode** drop-down list has been added to the **Hole** command. You can choose among **Full**, **Depth** and **Double**. The **Blind** value as **Extension** is available only when **Mode** is set to **Full** or **Double**.

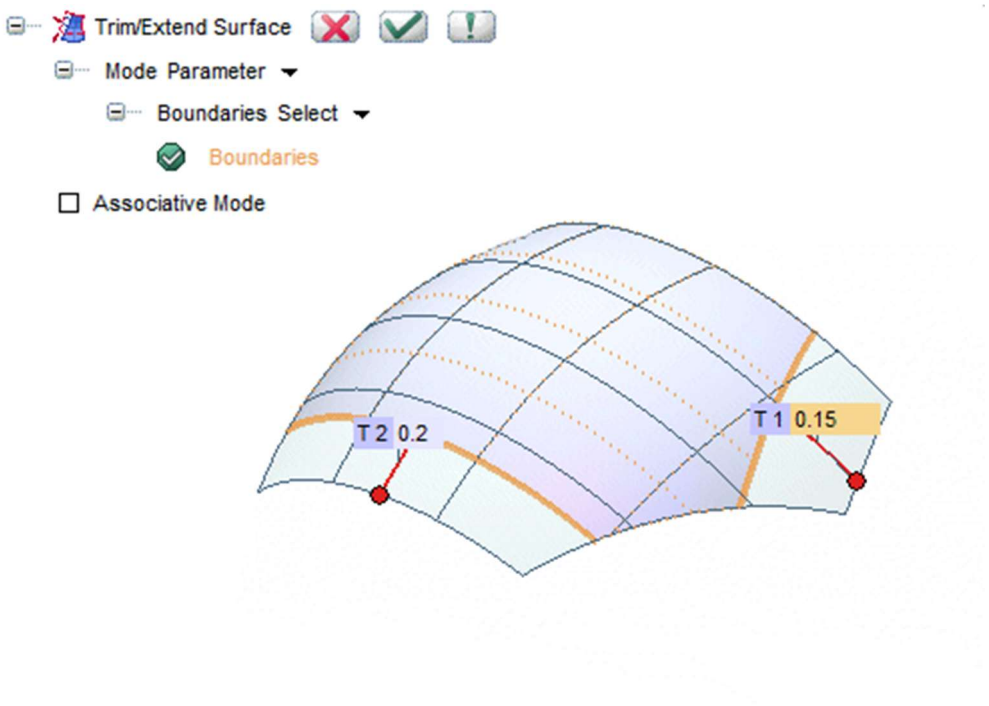
If you intentionally want to create a non-standard threaded hole (the hole diameter and thread values are not a standard combination), you have to create the hole by using the **Linear Slot** command and then apply the **Thread** command.

The internal **Undo/Redo** buttons have been added to the **Modify ⇨ Solid ⇨ Subdivision Solid** command to improve its usability.
When you change some setting within the command, the internal **Undo** button is available on top of the selection list to undo the changes. As soon as you click it, your changes will be undone and the **Redo** button will show up on top of the selection list. If you click it, your former changes will be applied again.

Surfaces

In the previous versions it was not possible to trim or extend a surface on its trimmed boundaries when the **Mode** drop-down list was set to **Parameter**.
Now the **Boundaries** drop-down list has been added to the **Parameter** mode: the value

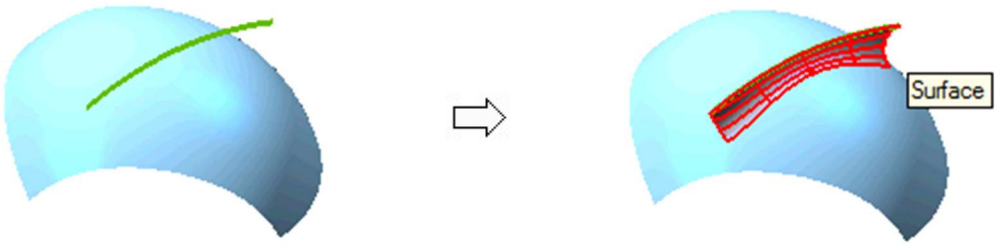
Not trimmed gives the same behavior as the previous versions; the value Select allows selecting any surface boundaries. In this second case, the parametric modification of the surface is performed modifying its trimming loops.



Another enhancement is related to the Double Fillet command. In the previous versions it was possible only to draw a fillet between two surfaces. Now with the new Mode drop down list, you are able to create the blend surface between a curve and a surface. You can select between:

Surface - surface It enables you to create a fillet or a chamfer between two selected surfaces.
When this mode is selected, the command works exactly like in previous version.

Curve - surface It enables you to create the blend surface between the selected curve and surface. The blend can be either rounded (fillet) or straight (chamfer). When the Fillet type is selected in the Type drop-down list, the Radius drop-down list is available, and you can choose among Constant, Variable, and Parabolic.
When this mode is selected, all the options under More Options are disabled and the Quadrant selector is not available, because it is useless.



In this command the **Restore Selection** modality is available for selections and settings.

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think3 Wizards (API-COM)

The think3 Wizards (used in the API-COM environment) are completely redesigned.
Now packages are available for the different Microsoft© Visual Studio® versions:

- Microsoft Visual Studio 2005
- Microsoft Visual Studio 2008
- Microsoft Visual Studio 2010
- Microsoft Visual Studio 2012

They are running in both the 32 bit and 64 bit environments.

Windows 8

The Windows 8 Operating System is now supported.

ThinkDesign 2014 Release Notes

Analysis

The new **Along Work Plane** check-box has been added to the **Analysis ⇨ Entity Size** command.

When selected, it allows to compute the bounding-box in the X, Y and Z directions relative to the current Work Plane. When not selected, the bounding-box is computed in the X, Y and Z directions relative to the World reference system.

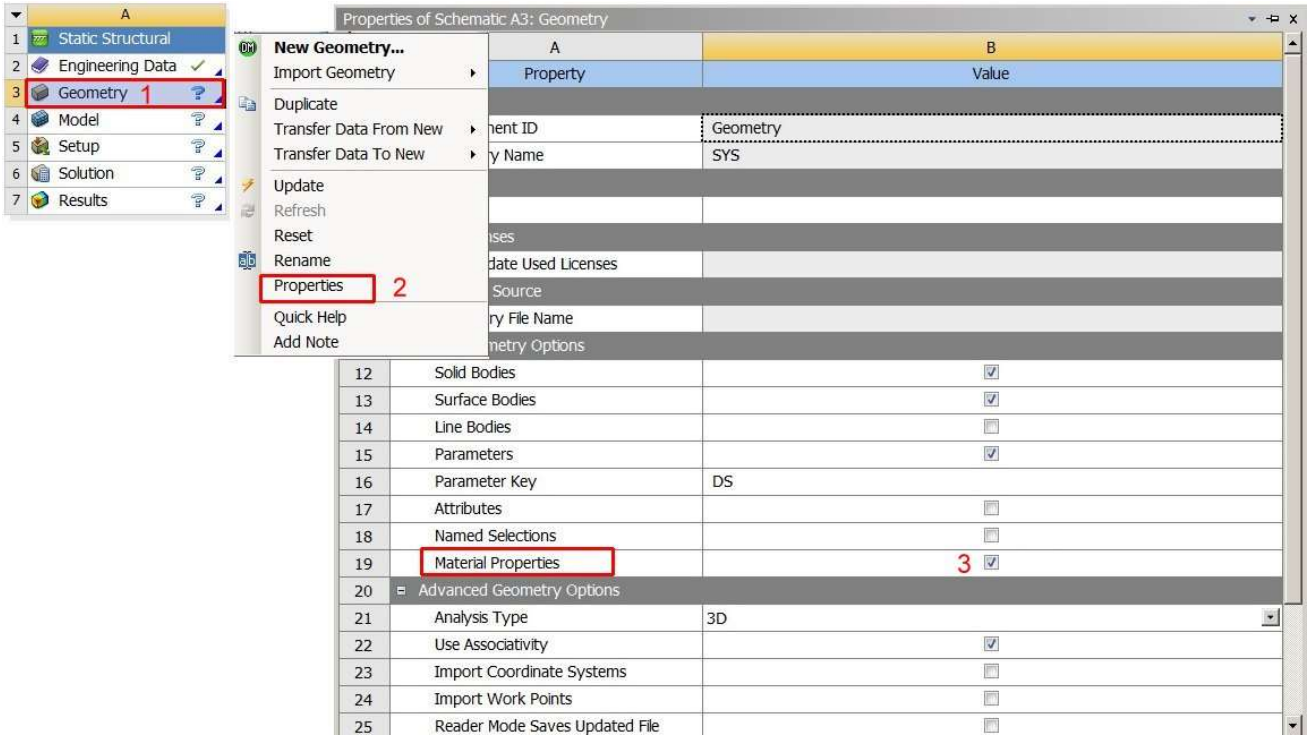
ANSYS plug-in

Choosing the ANSYS plug-in version

Starting from this version, you can choose whether to install either **ANSYS V14.5** or **ANSYS V15.0** plug-in from the custom window of the installation wizard during the installation process.

Improvements

In the previous versions of ThinkDesign, all the models imported in ANSYS had **Structural Steel** as **Material**.
From the 2014.1 version, instead, the **Material** associated to a solid in ThinkDesign is taken into account in ANSYS if it exists in **ANSYS Material Libraries** (name matching).
The correspondence is enabled only if **ANSYS Material Properties** option is checked-on.



In order to activate the option, follow these steps:

1. In the **Static Structural** menu, R-click on **Geometry**.
2. Choose **Properties**.
3. Select the **Material** check box.

Configuration Manager

After setting the **Smart Objects** and/or **Group** library path in the **Library Path** of the **System Options** dialog box, the new **Libraries** check box will be selectable in the **Configuration Options** dialog box of the **Configuration Manager ⇨ Save** command. Selecting it you can share also these settings with the other users.

Data Converter

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

- Any SolidWorks© version up to 2014; for Autodesk® Inventor® any version up to 2014
- Any JT version up to 9.5
- Any Siemens NXTM (Unigraphics) version from V11.0 up to NX 9
- Any CATIA® V5 version from R4 to V5-6R2013
- Any Parasolid® version up to v26.0
- Any Pro/ENGINEER version up to Wildfire 5.

For the Autodesk® DXF/DWG format, the DWGDirect 3rd party library has been upgraded to Teigha® 3.09 (compatible to library (Autodesk® DWG™ 2013 file format).

Improvements

A new category **Save-Visual Bookmark** has been added to the **Options** dialog box available in **Save as Adobe PDF file** (***.pdf**). The available options have the following meaning.

View Data	When selected, **Visual Bookmarks** containing viewport and clipping plane information are exported. When not selected, they are ignored.
Hide/Unhide Entities	When selected, **Visual Bookmarks** containing information on hidden and visible objects are exported. When not selected, they are ignored.

The **User Defined Properties** (UDP) assigned to the entire model or to entities are now exported into 3D PDF.
The new information in the PDF are visible in the **Model Tree Navigation Panes**.

MoldDesign

The **Mold Standard Parts** command has been improved in the selection of the **Positioning Points**.
After the selection of a positioning points, you can easily add to this selector other points by using the following context menu items:

Transform ⇨ Mirror about X axis	It allows you to add the mirrored points about X of the Work Plane.
Transform ⇨ Mirror about Y axis	It allows you to add the mirrored points about y of the Work Plane.
Transform ⇨ Copy along X direction	Through the **Copy of points** dialog box, it allows you to add a specific number of points along the X axis at a defined **Step Distance**.
Transform ⇨ Copy along Y direction	Through the **Copy of points** dialog box, it allows you to add a specific number of points along the Y axis at a defined **Step Distance**.
Transform ⇨ Reset Marker Selection	It allows you to reset the marker selection. For deselecting only one point, you have to click on it by pressing **<CTRL>**.

This enhancement allows you to position the standard parts in symmetric points along X or Y or create a pattern of standard parts along X or Y.
In addition, the **Prevent Rotation by Cutting** drop-down list has the new **Both sides - oblong** option. When it is selected, the profile you get is oblong instead of rectangular.
Multiple separate spacer components instead of one have been implemented in the **DME**, **HASCO** and **MEUSBURGER** catalogs.

In the previous version, when you inserted a Moldbase (DME, Hasco, Meusburger), the Guide pillar had all the same Diameter. In some case it is useful to have three guide pillars with same diameter and the 4th with another diameter. In order to allow this choice, the new **3 + 1** option for **Leader Pin Top Shank** and **Leader pin bushings** has been added to the **Guide Elements** tab of the **DME** and **MEUSBURGER** catalogs.
If the **3+1** option is **No**, all the four leader pins have same diameter. If it is **Yes**, three leader pins have same diameter and the 4th has a lesser diameter. The 4th leader pin will be placed at lower right of top view.
In order to access the new features in the **DME**, **HASCO** and **MEUSBURGER** catalogs you have to import the ones available in the 2014.1 version.

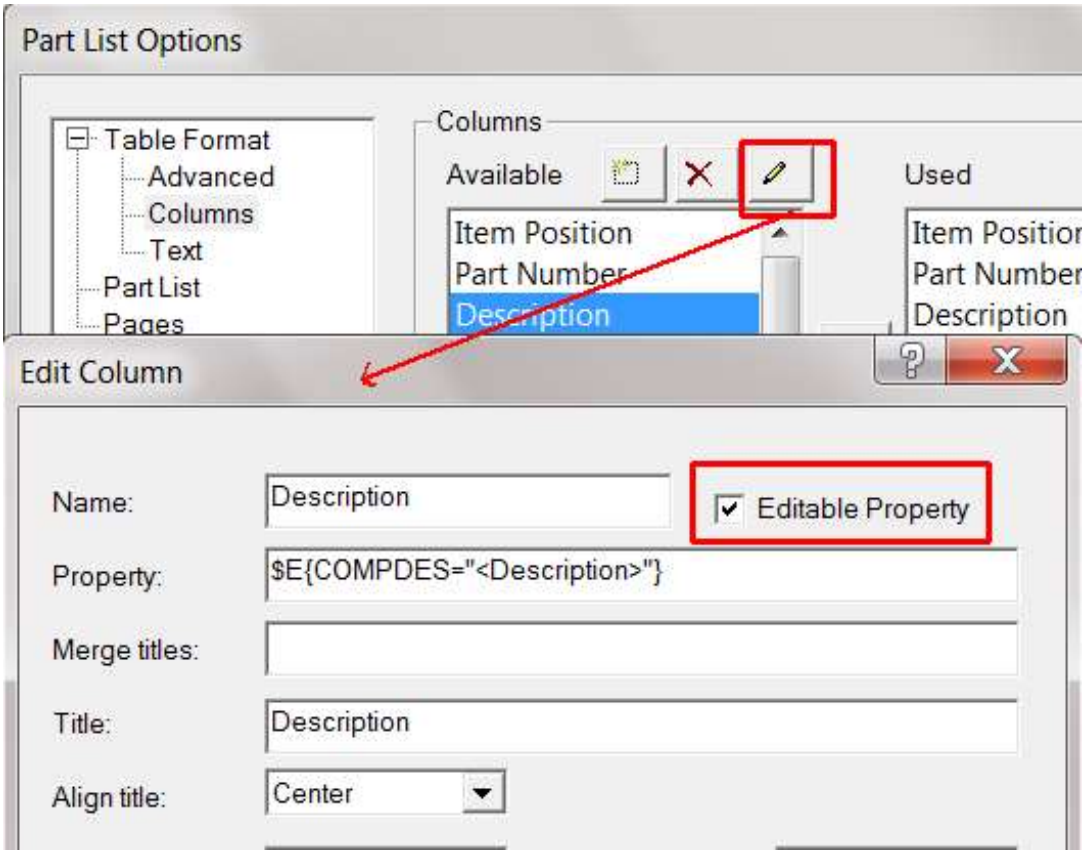
The **Part List Editor** has been improved for the MoldDesign user.
You can configure whether the column in the dialog box is editable or read only.
You can automatically calculate the size (bounding box) of a component and include it in the Part List.

PARTsolutions Plug-in

The **PARTSolutions Plug-in** has been updated to support the v. 9.04 of Cadenas PARTsolutions.

Part List Editor

The **Part List Editor** contains two extra columns, which show the **COMPONENT NAME** and the **HIERARCHY LEVEL** of the component in the assembly. These columns are fixed and not editable, moreover they are not considered when you **Export to CSV**.
Another important feature is the presence of the **Hide entities** and the **Unhide entities** commands in the context menu along with the **Zoom Entities** command; after selecting a component, they can hide/unhide the selected entities to allow you a better visualization of your model.
The new **Editable Property** check box allows to configure whether the column in the **Part List Editor** dialog box is editable or read only.



The editable columns have a white background, the others are grayed out.
Two new properties have been added among the ones available in the list of columns in the **Table Format** - **Columns** category of the **Part List Options** dialog box:

CompSizeAuto It is a checkbox in part list editor and determines whether the size has to be automatically calculated. When selected, the program automatically calculates the size of the component; the content of the corresponding field of the **CompSize** column is not editable. When cleared, the string in the corresponding field of the **CompSize** column is editable. By default the check box is selected.

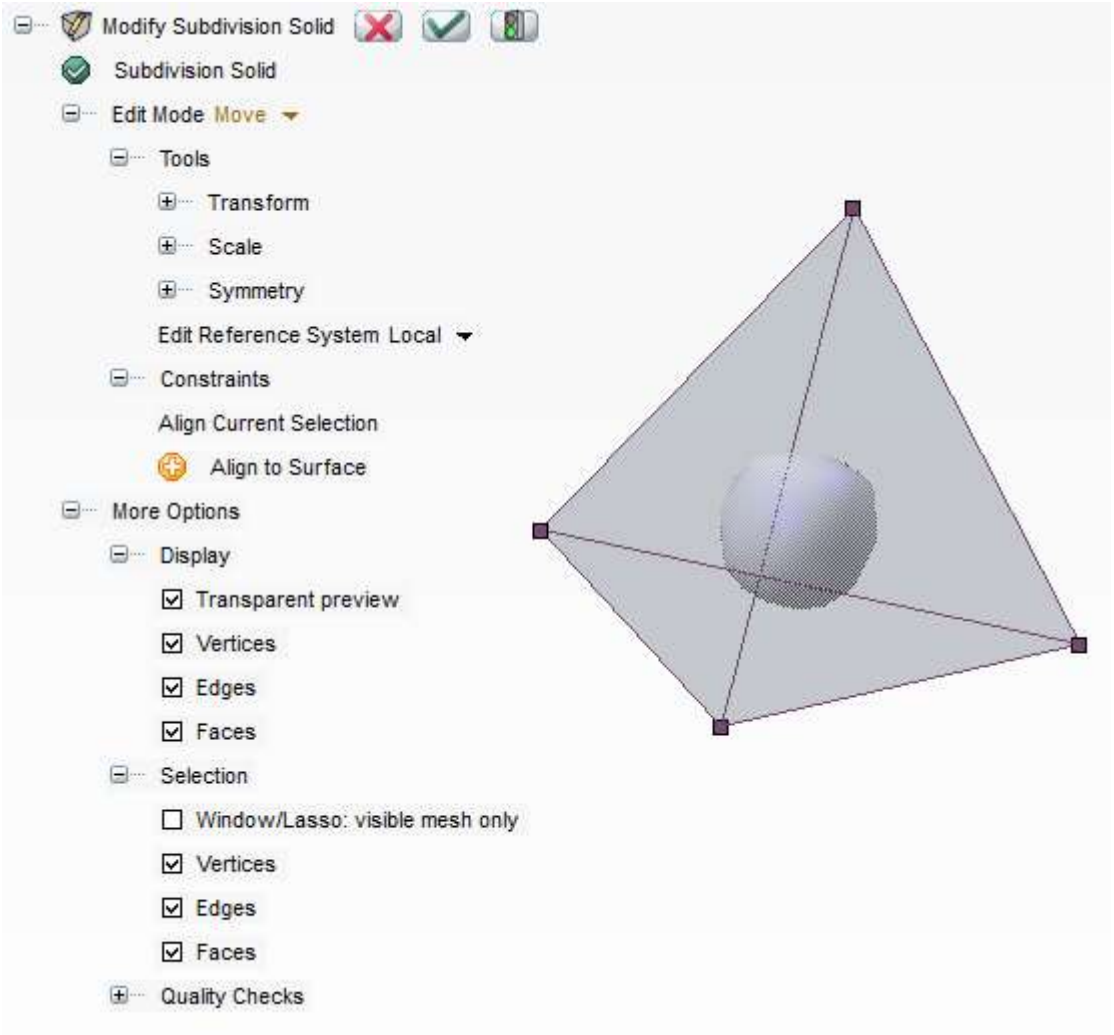
CompSize It is a string like 142L x 82W x 17H (length x width x height). The automatic value is the bounding-box of the component with respect to the axis of the component itself.

Performance Improvements

In this version several improvements have been made to increase the performance of ThinkDesign graphical feedbacks.
When switching between model and drawing environment the visualization of the graphical items, like toolbars, is faster.
The time of displaying the command selection list has been significantly reduced.

Subdivision Solid

The **Subdivision Solid** command has been improved, in terms of usability and functionality. First of all, it has been provided with a new user interface, that is more intuitive, ergonomic and efficient.



New and simpler selection methods have been developed by introducing the standard use off markers. When the mouse is placed over a face, edge or vertex, it is highlighted in the default ThinkDesign mouse-over color (namely red). When the marker is clicked upon, it is highlighted (in orange by default) and selected. Holding the **<CTRL>** key and clicking on an item will deselect it if it was in the selection, otherwise it will add it to the current selection. **Window** and **Lasso** selection are also enabled. Furthermore, whenever a current selection is available, an item is added to the context menu (accessed by right-clicking anywhere in the drawing area) called **Reset Current Selection**, so as to enable you to empty the current selection.

Edit Mode Changes

Split face, **Remove face** and **Subdivide face** modes remain unchanged apart from the

selection being made directly onto the face markers. Move and Extrude face modes have been modified and a new Add face mode appears in the drop-down list.

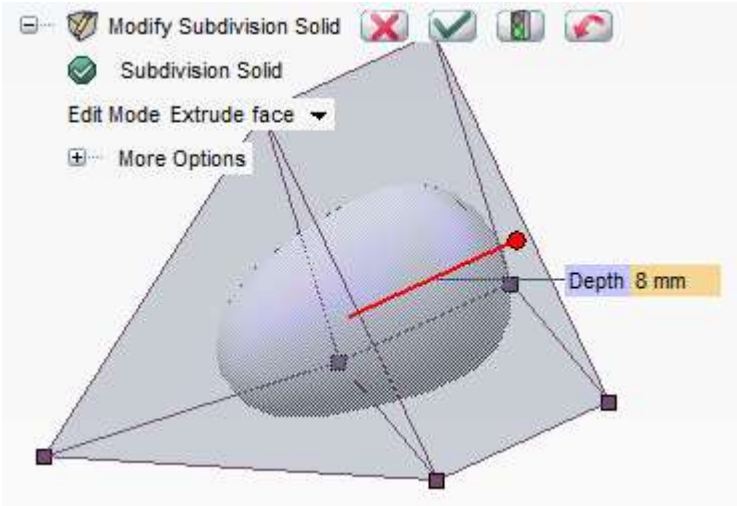
Move

For Move operations, there remains only one mode (compared to the 3 in the previous version) in which any selected markers can:

1. Be dragged freely with the mouse, following the dragging method specified in the software input mode (modified in Edit ⇨ Input or in System Options ⇨ Input ⇨ Mouse Input)
2. Be dragged along certain axes after pressing the same keys as in Control Point Editing (in the work plane: 'x' for the X-axis, 'y' for the Y-axis, 'w' for the Z-axis, and along the screen: 'v' for the vertical axis, 'h' for the horizontal and 'd' for the depth, this last one being mapped along the horizontal). Note that to leave this mode either <ESC> or the last key is pressed again. Use the Transform tool to move the selection more precisely.
3. Use the Scale tool to stretch the selection along specified axes.
4. Use the Symmetry tool to specify a symmetry plane and move the selection along with its symmetrical equivalent, if any, in mirror motions with respect to the chosen plane.

Extrude face - Depth Handle

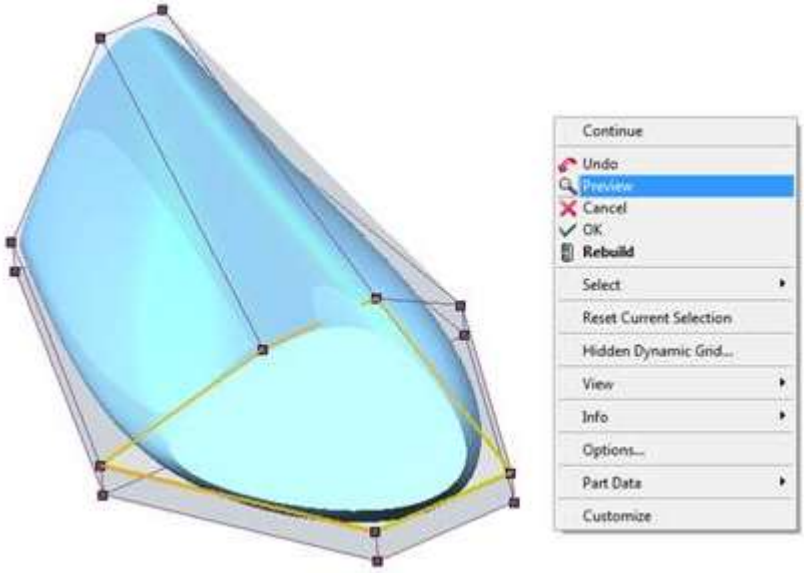
Extrude face has been enhanced with a depth handle for the user to control the length at which the extrusion is made. Simply select the face to extrude and the handle will appear at the default length (that which was previously used). The handle can then be dragged, or the depth can be entered manually to get the desired shape. Do note that as before, if during the extrude operation two faces coincide exactly, that is their vertices match, the faces will be destroyed, and the shape fused on these vertices.



Add face

A new edit mode is now available to add a face. It is a counterpart to the Remove face

mode. If the selection of edges is valid, the standard preview button appears at the top of the selection list. When this is clicked, the corresponding face will be built and the underlying solid updated. Do note that the added face will be the smallest possible given the selection as illustrated below.



More Options

This menu is shared by every mode. It contains utilities for the selection and display of markers and Quality Check for the subdivision solid surfaces. This last one remains unchanged from the previous version.

Windows 8.1

The Windows 8.1 Operating System is now supported on the standard PC platforms.

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think3 Wizards (API-COM) Support

The think3 Wizards (used in the API-COM environment) is now available for Microsoft Visual Studio 2013.

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

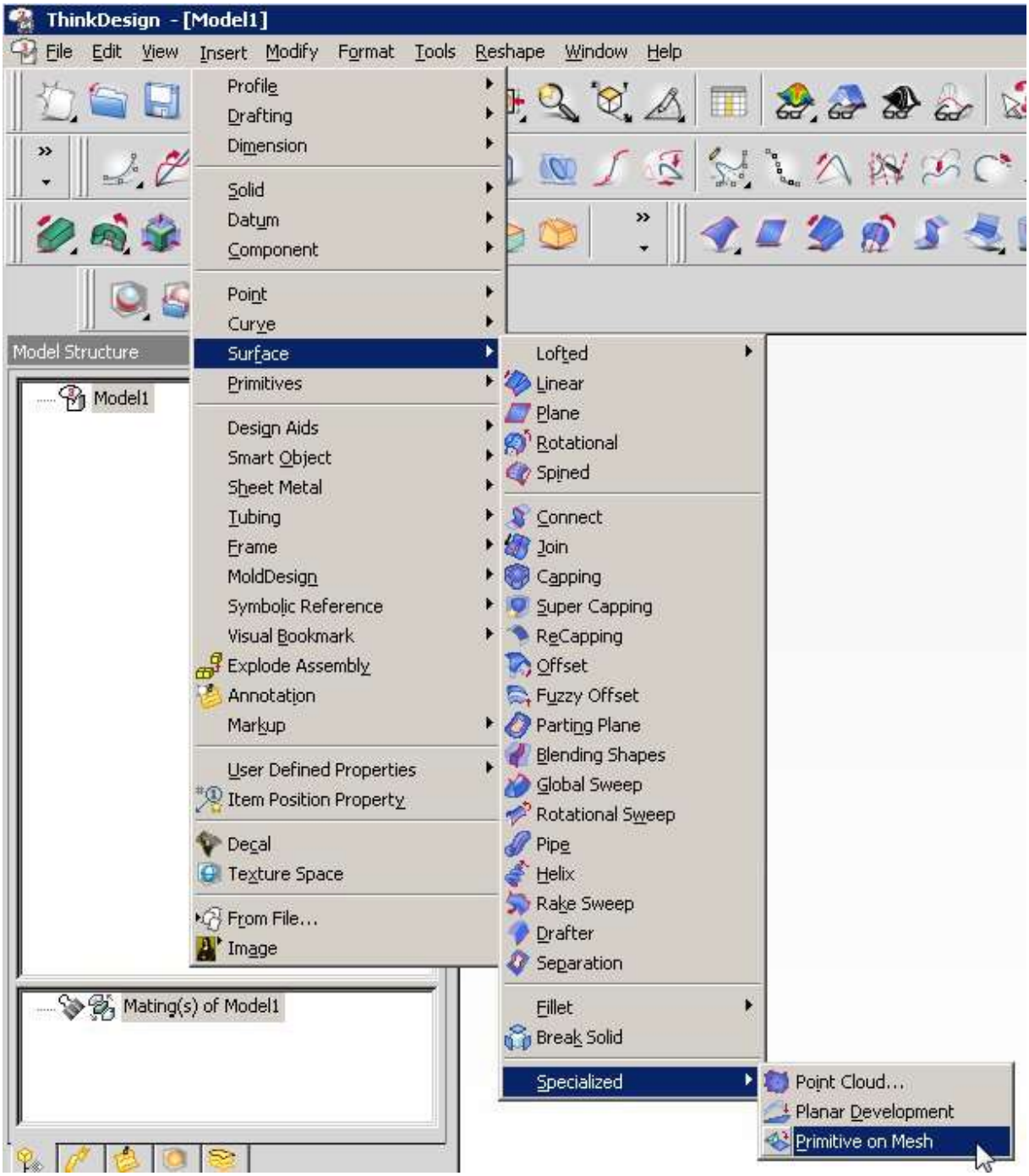
Starting from this version, beyond versions 14.5 and 15.0, you can also choose whether to install either ANSYS V16.2 or ANSYS V17.0 plug-in from the custom window of the installation wizard during the installation process.

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

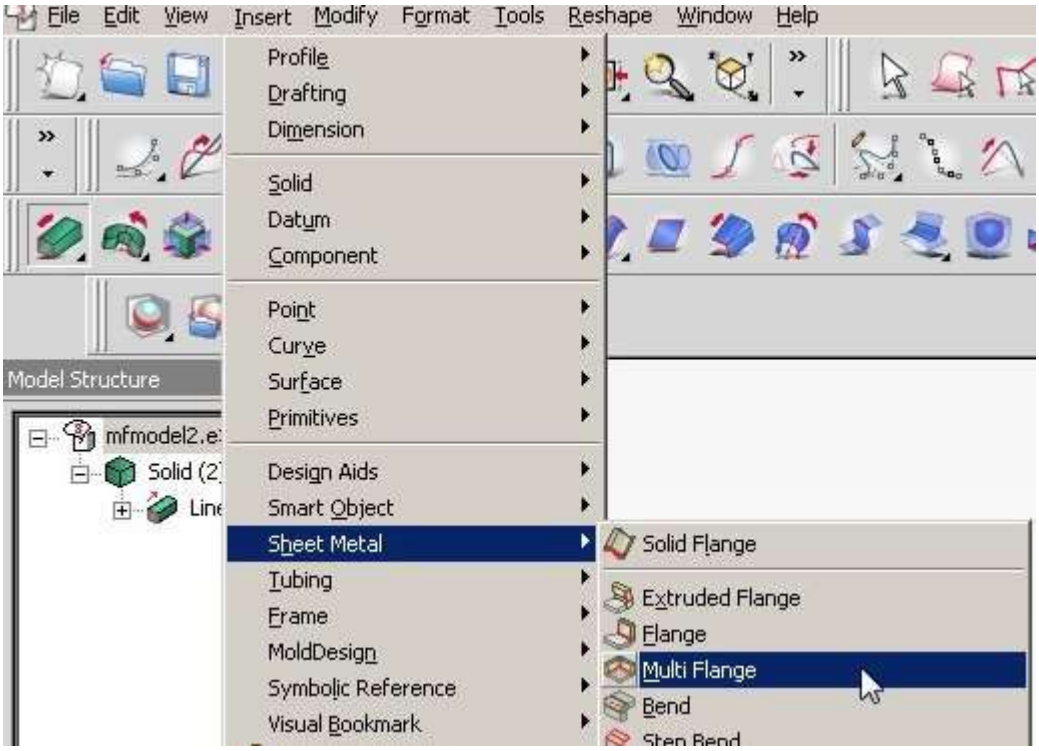
as criteria the angle between the normal of the picked facet and the normal of the other facets of the mesh.

- 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

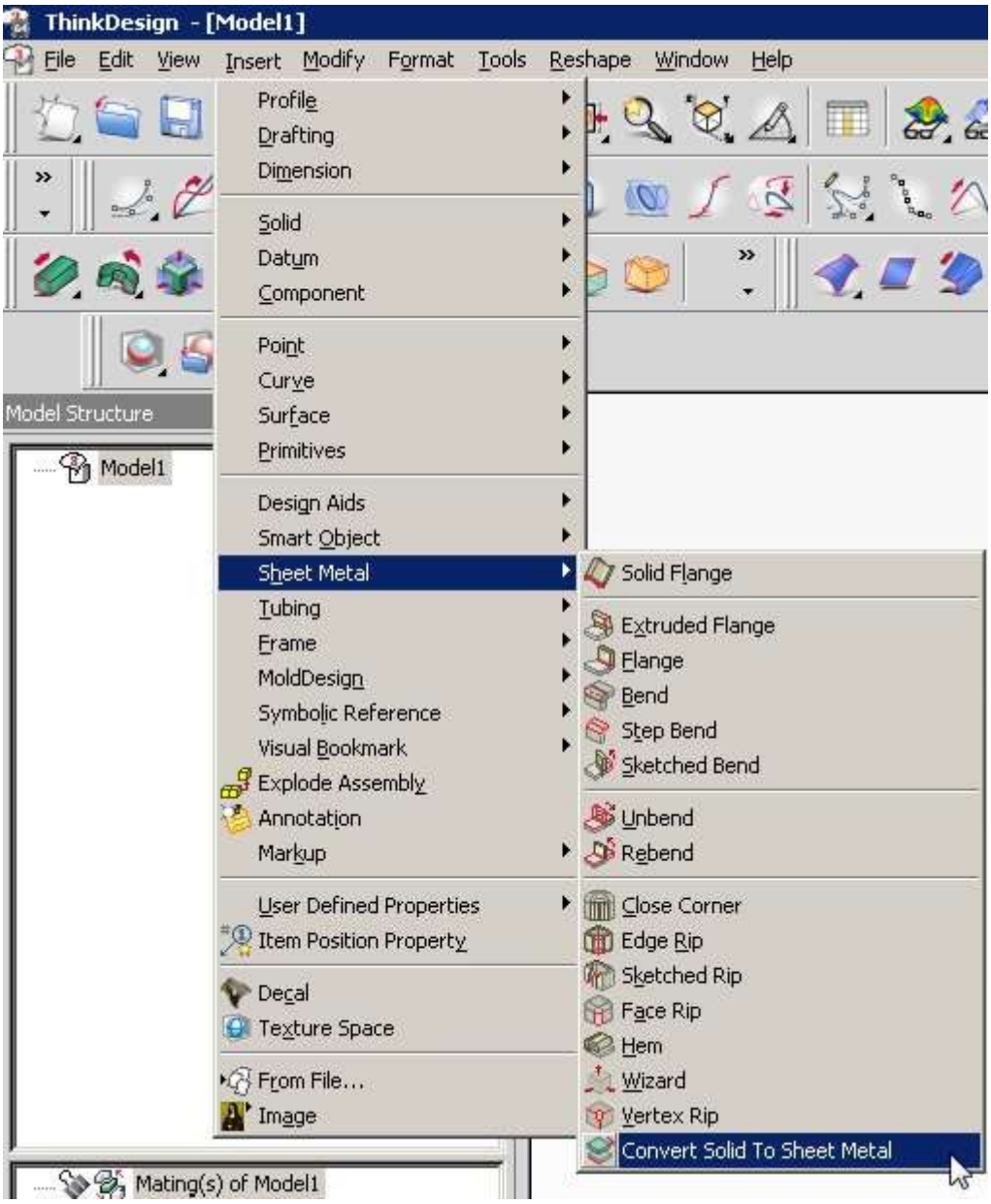
The new **Multi Flange** command enables you to create multiple flanges within the same command session.



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.

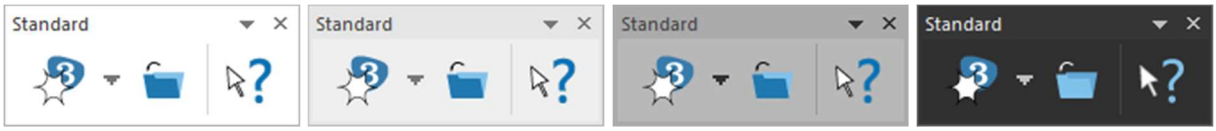
ThinkDesign 2017 Release Notes

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:

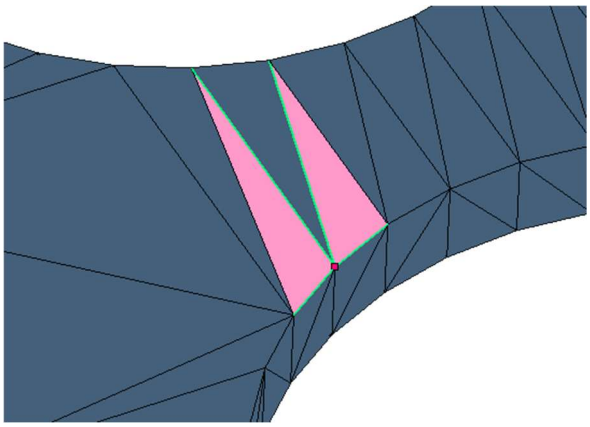
- New “Mesh Repair” command
- New “Remove Facet” command
- Enhancements in View and Zebra commands using meshes
- General improvement of reliability and robustness in the management of poor quality meshes

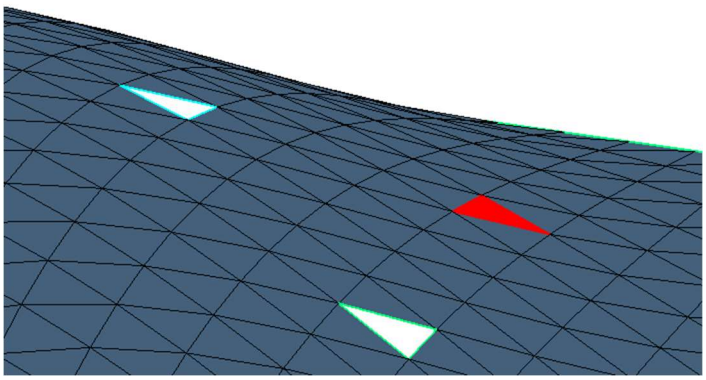
New “Mesh Repair” command (Modify ⇨ Mesh ⇨ Repair)
This command enables the user to check the quality of a given mesh and repair its defects.
With improved quality meshes, commands such as **Compensator**, **Super Capping** or **3D Printing** are more reliable and efficient.

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:

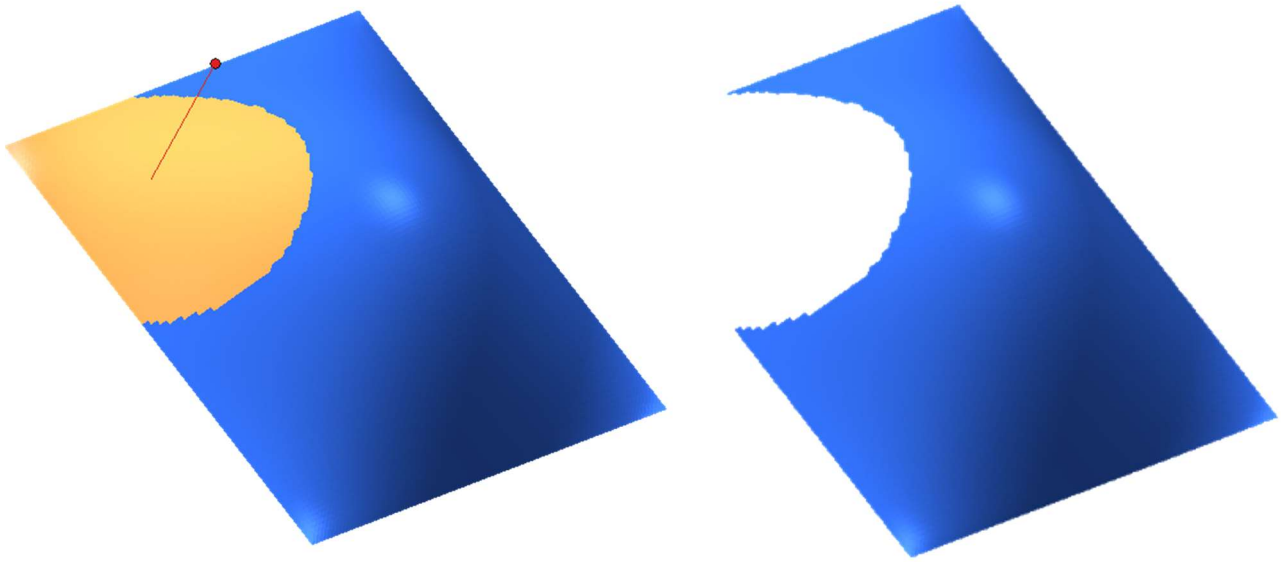
- Change the color of display of these defects using the color button,
- Check the box to visualize and repair these defects: the number of defects is displayed as well as a **Repair all defects** button. If there is no defect to repair, the button is grayed,

- 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

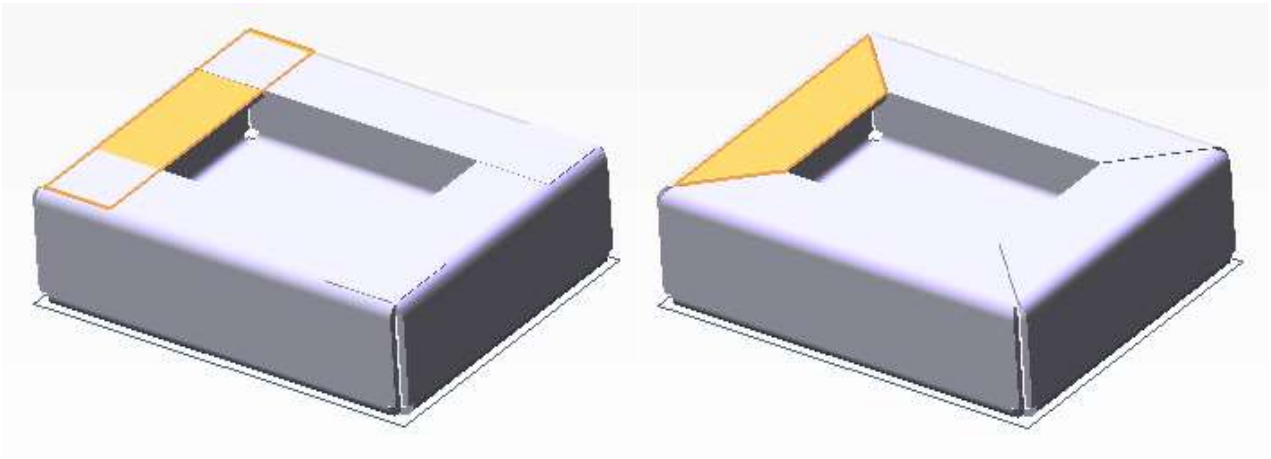
The **TD XchangeReader** module is now capable to read:

- 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.