ThinkDesign 2019 Release Notes

Many improvements have been made in version 2019. This document contains a list of short descriptions for each new feature, improvement and enhancement.

Full Hybrid Modeling

ThinkDesign 2019.1 introduces some major changes for the Full Hybrid Modeling:

- Reshape functionalities re-dispatched in standard ThinkDesign menus
- New commands
  - Separate Mesh
  - Join Mesh
  - Offset Mesh
  - Mesh From Solid
  - Invert Mesh
  - Split Edges
  - Swap Edges
- Enhanced commands
  - Stanford Polygon Format (PLY) loading and saving
  - Unit of measurement for loading STL, OBJ, PLY
  - Split Mesh with Plane
  - Split Mesh with Curves
  - Refine Mesh
  - Analysis Normals
  - Boundary Curve
  - Point on Intersection of Curves/Surface/Mesh
  - Primitive on Mesh

Reshape functionalities re-dispatched in standard ThinkDesign menus

ThinkDesign has support for the Reverse Engineering activity with the **Reshape** and **HQRE** modules, including functionalities to manage meshes.
With previous releases, the user had been accessing **Reshape** commands from a separated Reshape drop-down menu in the menu bar.
Starting from ThinkDesign 2019.1, the **Reshape** commands, just like HQRE ones, are re-dispatched in respective places in menus, **Insert**|**Modify** ⇨ **Curve**|**Surface**|**Mesh**. The activity toolbars Reshape and HQRE are still there.

Some commands have been moved to the standard menus **File**, **Insert** and **Modify**:

Reshape command	Moved to	Note
Split Mesh with Plane	Modify ⇨ Mesh ⇨ Split with Plane	Enhanced
Split Mesh with Curve	Modify ⇨ Mesh ⇨ Split with Curves	Enhanced
Smooth Mesh	Modify ⇨ Mesh ⇨ Smooth	
Decimate Mesh	Modify ⇨ Mesh ⇨ Decimate	
Refine Mesh	Modify ⇨ Mesh ⇨ Refine	Enhanced
Sew Mesh	Modify ⇨ Mesh ⇨ Sew	
Remove Wrong Faces	Modify ⇨ Mesh ⇨ Remove Wrong Faces	
Fill Mesh Holes	Modify ⇨ Mesh ⇨ Fill Holes	
Curve on Mesh	Insert ⇨ Curve ⇨ Curve on Mesh	
Mesh Character Lines	Insert ⇨ Curve ⇨ Mesh Character Lines	
AutoPatching	Insert ⇨ Surface ⇨ Specialized ⇨ AutoPatching	

Some others are covered by existing or new commands:

Reshape command	Covered by	Note
Import Mesh	File ⇨ Open... or Insert ⇨ From File... *.STL, *.PLY, *.OBJ,...	Enhanced
Export Mesh	File ⇨ Save or File ⇨ Save As... *.STL, *.PLY, *.OBJ,...	Enhanced

Cut Mesh	Modify ⇨ Mesh ⇨ Remove Facets (option **Split** removed) Insert ⇨ Mesh ⇨ Separate (for the option **Split**)	Modified New
Heal Mesh Normals	Modify ⇨ Mesh ⇨ Repair (to fix orientation issues) Modify ⇨ Mesh ⇨ Invert (to invert the normals) Tools ⇨ Info ⇨ Analysis ⇨ Normals (to visualize the normals)	New
Mesh Boundaries	Insert ⇨ Curve ⇨ Boundaries	Enhanced
Plane on Mesh	Insert ⇨ Surface ⇨ Specialized ⇨ Primitive on Mesh	

Other commands have also been re-dispatched:

Compensator command	Moved to	Note
Offset Mesh	Insert ⇨ Mesh ⇨ Offset	

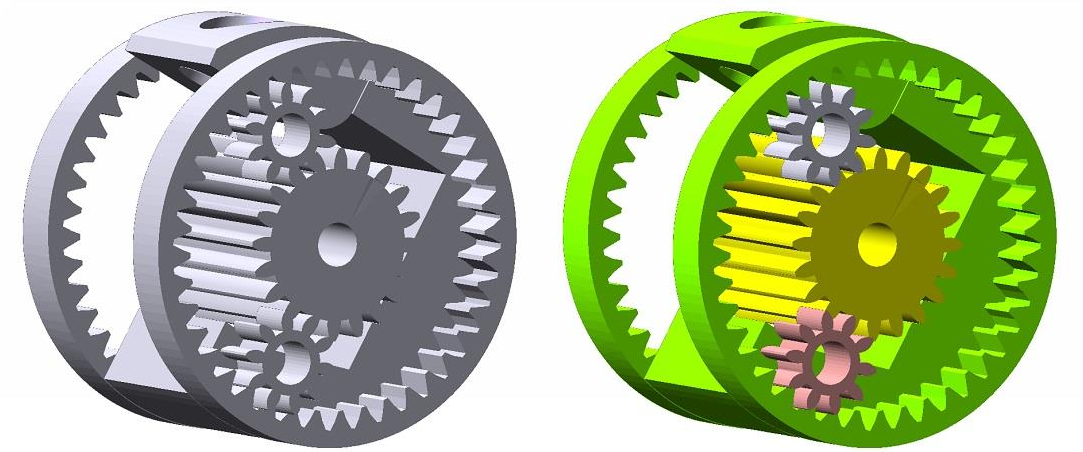
File ⇨ 3D Printing ⇨ Create Mesh command	Moved to	Note
Create Mesh	Insert ⇨ Mesh ⇨ From Solid	

New commands

Separate mesh command (**Insert** ⇨ **Mesh** ⇨ **Separate**)
This command enables the user to *separate* a triangular mesh in different parts, by selection or by disconnected shells.
In the **Mode** drop-down list, choose the way the mesh will be separated:

- **Selection** — the selected facets will be grouped in a new mesh and removed from the initial one.
- **Shells** — the selected mesh will be separated: one mesh for every disconnected shell (set of connected facets) is created.

The command can assign automatically a different layer and color for each created Mesh part. In the **Layer increment** and **Color increment** boxes under **More Options**, you can specify the increase in layer and color between a part and the next (the layer is the one displayed in the **Layers** tab; the color number is the one displayed in the color palette).



Initial mesh *Separated meshes*

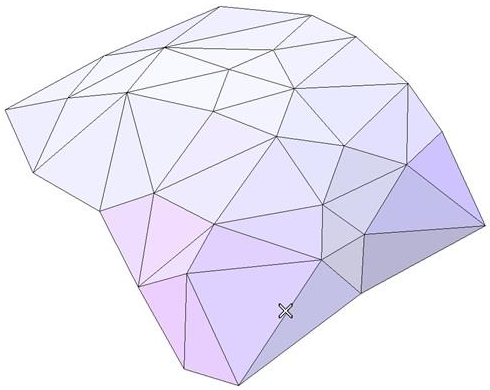
Join Mesh command (**Insert ⇨ Mesh ⇨ Join**)
This command enables the user *to join* several triangular meshes to create a single mesh. If selected meshes are disjoint, the resulting mesh will be multishell. Nodes closer than the specified **Merge tolerance** value are merged together.

Offset Mesh command (**Insert ⇨ Mesh ⇨ Offset**)
The command has been moved from **Modify ⇨ Compensator** to **Insert ⇨ Mesh** menu. The functionality remains unchanged.

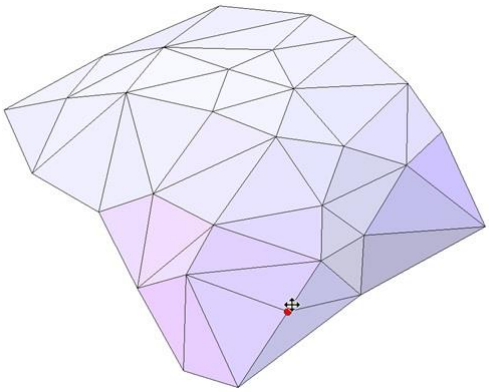
Mesh From Solid command (**Insert ⇨ Mesh ⇨ From Solid**)
This command enables the user *to convert solids to meshes*.
It replaces the mode **Create Mesh** moved from **3D Printing** command.

Invert Mesh command (**Modify ⇨ Mesh ⇨ Invert**)
This command enables the user *to invert the normals to the facets* of a triangular mesh.
In the **Invert Normals** drop-down list, choose the way the mesh will be inverted: **Manually**, **By direction**, **By axis** or **By point**.

Split Edges command (**Modify ⇨ Mesh ⇨ Split Edges**)
This command enables the user *to split the edges* of a triangular mesh. The original edge will be cut. The inserted node is connected to the opposite nodes in the adjacent triangles. It is useful for improving local mesh quality.

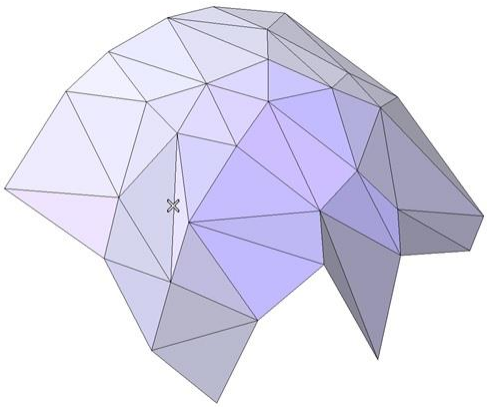


Pick the edge to split

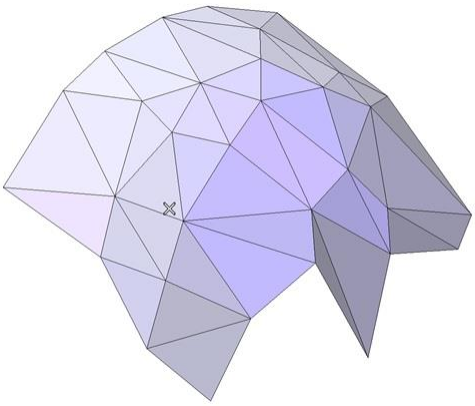


Drag the handle to move cut location

Swap Edges command (**Modify ⇨ Mesh ⇨ Swap Edges**)
This command enables the user *to swap the edges* of a triangular mesh. The original edge will be replaced by an edge that connects the other two corners of the adjacent triangles. It is useful for improving local mesh quality.



Pick the edge to swap



Swapped edge

Enhanced commands

Stanford Polygon Format (PLY) loading and saving

The PLY format, also known as the Stanford PoLYgon Format, is commonly used to exchange meshes.
ThinkDesign 2019.1 enables reading and writing files in the PLY format using respectively **Open/Insert ⇨ From File...** and **Save/Save As...**.

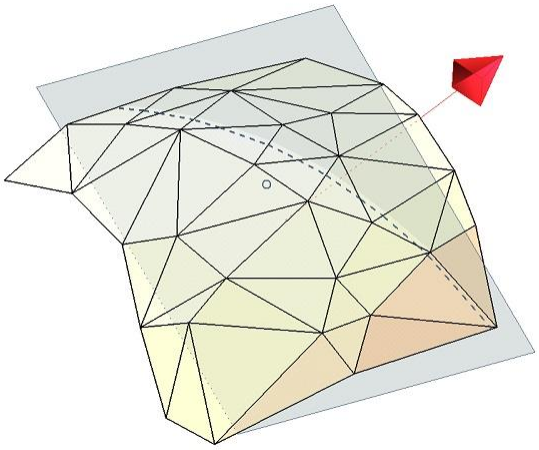
Unit of measurement for loading STL, OBJ, PLY

The file formats STL, OBJ, PLY do not describe a unit of measurement in their specification.
In ThinkDesign 2019.1, on **Load** option pages for the aforementioned formats, the user

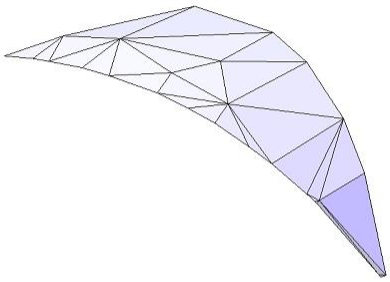
can select the unit that he wishes to interpret the imported data from the **Units of measurement** drop-down list.

Split Mesh with Plane command (**Modify ⇨ Mesh ⇨ Split with Plane**)
Like some other commands dealing with meshes, this command has evolved for better performances and usability.
The user interface changes slightly:

- Selecting multiple meshes is allowed.
- The **Split** option has been renamed as **Keep both sides**. The functionality remains the same.



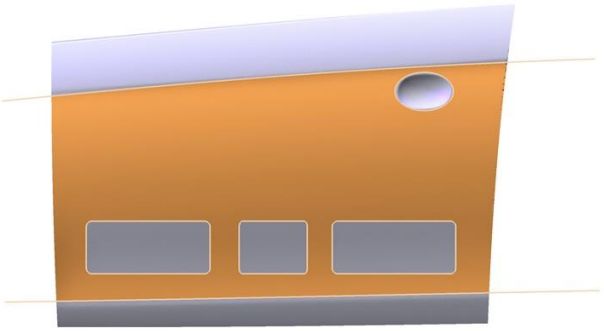
Initial mesh, limiting plane



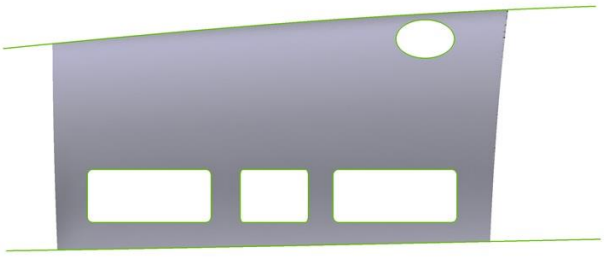
Same mesh after splitting

Split Mesh with Curves command (**Modify ⇨ Mesh ⇨ Split with Curves**)
This command has been greatly enhanced:

- The usability is much better:
  - The way to retain some split parts is clear and easy with the **Regions to Keep** selector.
  - It is not necessary anymore the limiting curves to form a closed loop or to be a single chain.
- The quality of the result, robustness and reliable, the performances are highly improved.



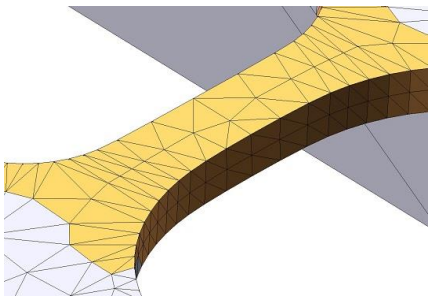
Initial mesh, limiting curves and region to keep



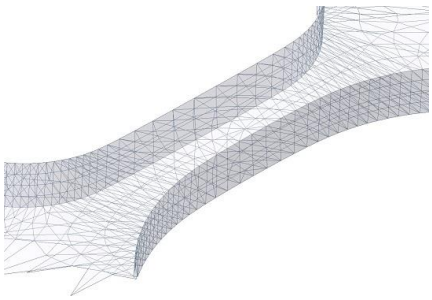
Same mesh after splitting

Refine Mesh command (**Modify ⇨ Mesh ⇨ Refine**)
The improvements impact:

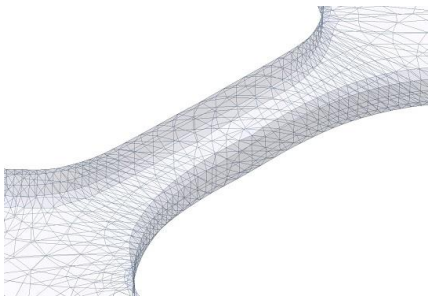
- The usability: to refine only a portion of the mesh, select the **Mesh Zones** check box and subsequently select the mesh facets to be included for refining. Then the facets can be selected either one by one by picking them or using more convenient tools:
    - Window or Lasso selections with **Select Facets in Front Only** option ON or OFF.
    - Smart selection commands dedicated to facets of mesh (Crease Angle and Geodesic Distance).
- The capabilities: some additional settings can be done under the **Mesh control** node:
    - A **Crease detection** value enables the user to preserve or not creases on the mesh.
    - A **Corner detection** value enables the user to preserve or not corners on the mesh.



Mesh zone to refine



Crease detection = 150°



Crease detection = 10°

Analysis Normals command (**Tools ⇨ Info ⇨ Analysis ⇨ Normals**)
Previously, when the selected shape was a mesh, the command displayed the normals to the facets. Now, a new drop-down list has been introduced to display the normals either **To Facets** or **To Nodes**.

Boundary Curve command (**Insert ⇨ Curve ⇨ Boundaries**)
Starting from ThinkDesign 2019.1, it is possible *to create the boundary curves* of triangular meshes. The resulting curves are polylines (degree 1 NURBS).

Point on Intersection of Curves/Surface/Mesh command (**Insert ⇨ Point ⇨ On Intersection of Curves/Surface/Mesh**)
Starting from ThinkDesign 2019.1, the command **Point on Intersection of Curves/Surface**, now labelled **Point on Intersection of Curves/Surface/Mesh**, enables you to select a mesh in the **Surface/Mesh** selector and to compute intersection points between the curves and the mesh.

Primitive on Mesh command (**Insert ⇨ Surface ⇨ Specialized ⇨ Primitive on Mesh**)
The command shows the optimum value of the cylinder radius when it is not constrained. This can be very useful to better control the shape to build.

Sheet Metal

Convert solid to sheet metal, Fixed face definition (**Insert ⇨ Sheet Metal ⇨ Convert solid to sheet metal**)
While converting a solid to sheet metal, the command now allows to select a fixed face, otherwise defined automatically. This fixed face is the one on which the Flat Pattern View should be created.

Organize Formats

Organize Formats (Format ⇨ Organize ⇨ Formats)
Material attribute can be now managed in General category like Layer or Color attributes.

Data Converters Upgrades

The TDXchangeReader module is now capable to read:

ThinkDesign 2019.1.SP2:

- Any CATIA® V5 version up to V5-6 R2018 (R28) (Enhancements in terms of bug fixes)
- Any SolidWorks© version up to 2019
- Any Parasolid® version up to v31
- Any Creo version up to 5.0
- Any JT version up to 10.2

General Software Improvement and Better Reliability

The source code has been accurately revised so that the program has highly improved its robustness and its reliability has been made much higher.