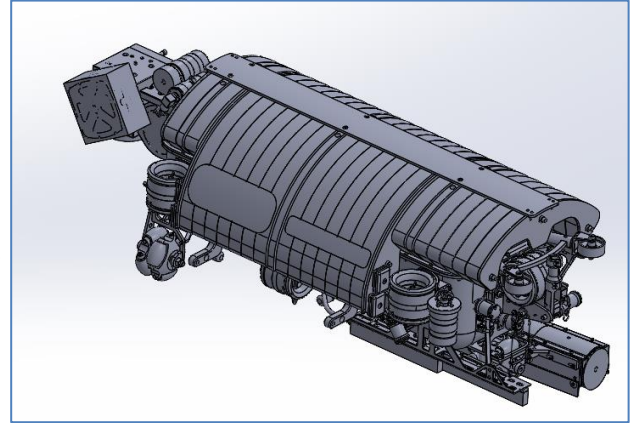


Handling Imported Files in SOLIDWORKS

SOLIDWORKS supports various file formats to import the data into SOLIDWORKS. In this, we can see about how to handle the Importing files from other design platforms for seamless communication and Performance.

To know more about SOLIDWORKS Supported File Formats: [Click Here](#)

Let us take an example of the model (Square Robot) for reference.



Neutral Files:

STEP (.stp), IGES (.igs), Parasolid (.x_t)

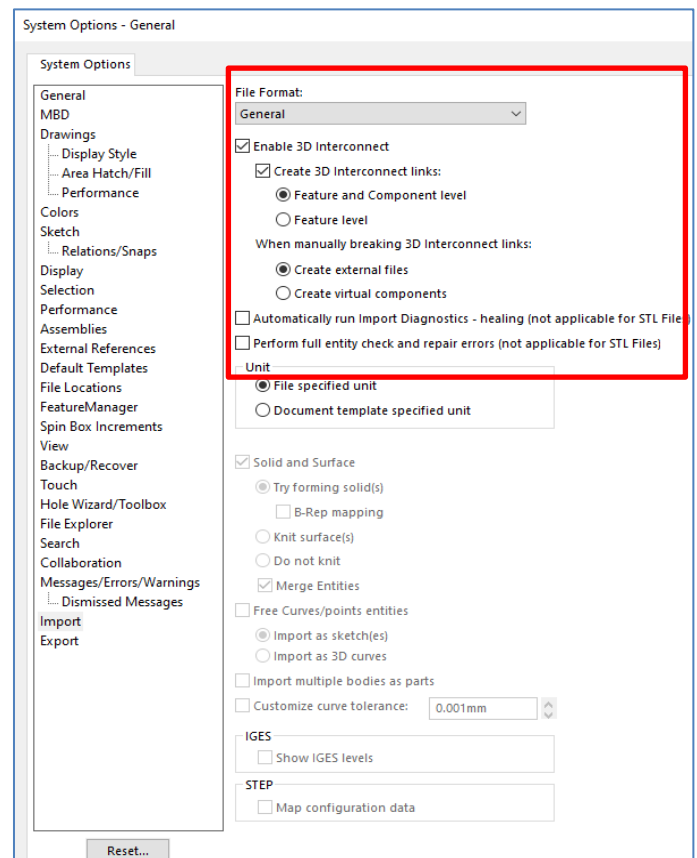
- These file formats are the typical import/export method for bringing files into and out of SOLIDWORKS.
- To improve performance in handling STEP files, follow the methods suggested below

Importing Options

Modify the Import options as per the below image. It applies to all imported files in SOLIDWORKS.

3D Interconnect: (Since SOLIDWORKS 2017)

- Click "Enable 3DInterconnect", which used to Replace SOLIDWORKS current translation Capabilities with new technology and workflows for working with third-party CAD data.
- Once the file is imported, Break link between the inserted STEP and original file.
- By breaking the link, the inserted part feature Dissolves and the part retains all the folders and sub-folders. All inserted part features become regular SOLIDWORKS features that you can modify or delete.



- Uncheck "Automatically run Import Diagnostics"

To avoid Diagnostics while importing, Uncheck "Perform full entity check and repair errors" due to the Import performance is slower because the software spends time checking and repairing the model entities.

Settings for STEP Format

Entities to Import

- ✓ Solids and Surfaces

In Options select,

- ✓ Import multiple bodies as parts

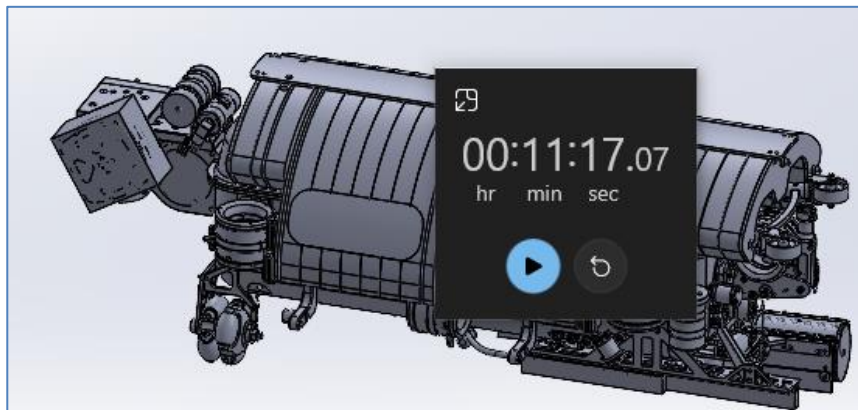
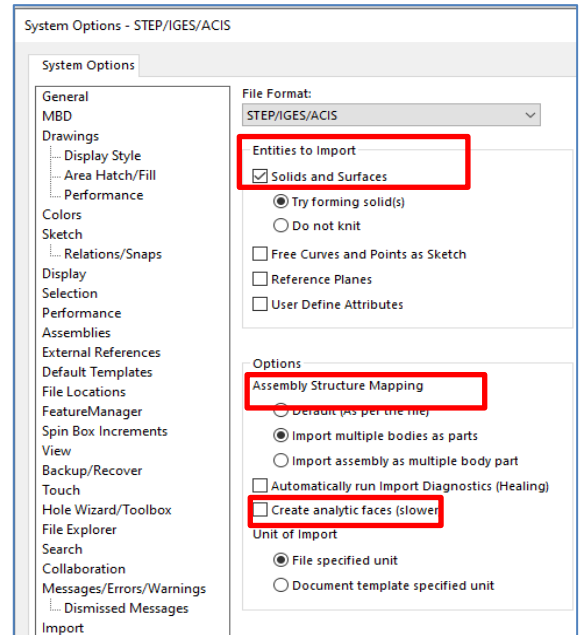
Unit of Import

- ✓ Use "File specified unit"

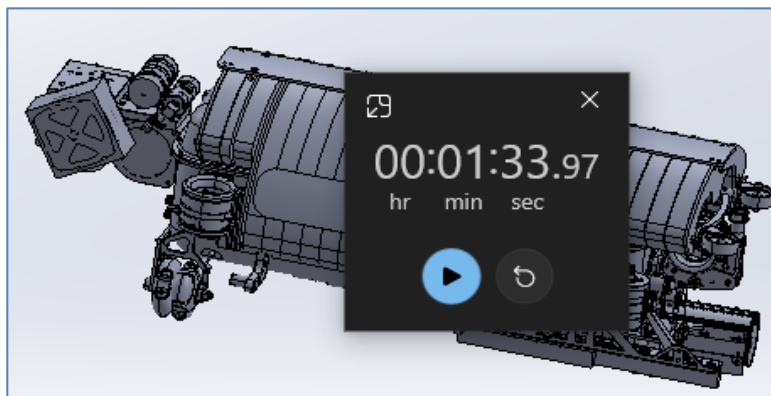
After modifying the Import Settings, we can identify the difference in time taken to opening files.

Let us work with STEP file consists of 2322 Components.

From the below image, we can see the differences.



Without 3DInterconnect and default Import Settings



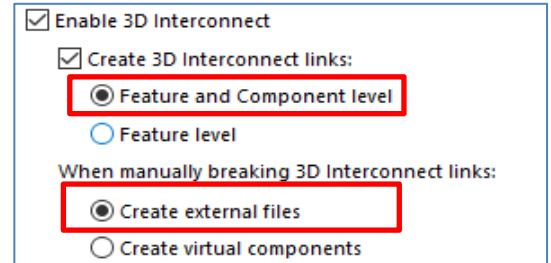
With 3DInterconnect and recommended Import settings

Once the file was imported in SOLIDWORKS, break the link with original file.

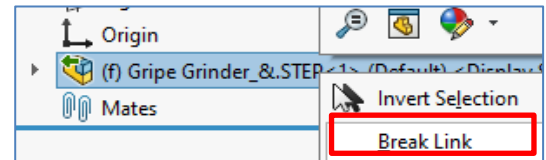
In latest version of SOLIDWORKS, some additional options that gives user more power to decide what to do with imported parts and assemblies.

Create 3DInterconnect links: Creates a link between the imported file and the original file.

Feature and Component level: When setting is enabled, imported components & sub-assemblies Will be imported with active link to original file.



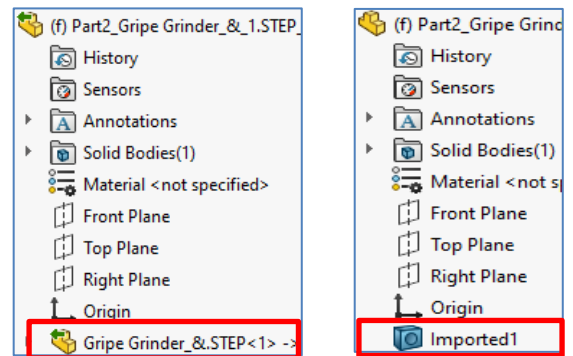
Feature level: When setting is enabled, imported assemblies are always imported with broken link to original file. Imported parts will be imported with active link to original file.



Manually breaking 3DInterconnect links: After the file imported, we can manually break the link between the imported file and the original file by selecting the option "Break Link".

Create external files: When setting is enabled, imported assemblies and their components will be imported as external files.

Create virtual components: In this setting, assembly Components will be imported as virtual components (they only exist in assembly and they don't have physical file).



(Before)

(After)

After breaking the link, save the file to native file format

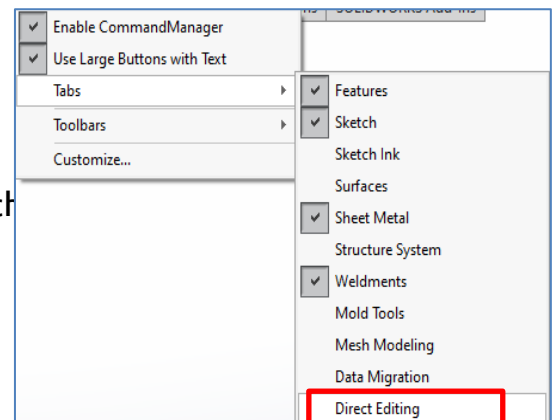
(e.g.: .sldasm)

Modification on Imported Data using Direct Editing

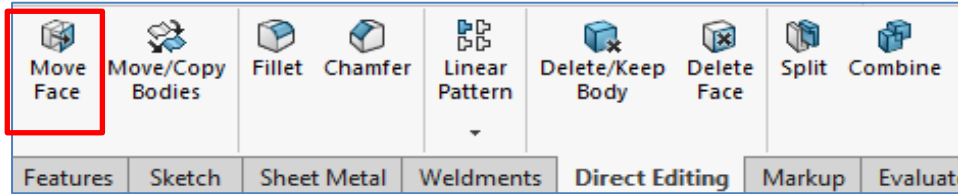
To modify the geometry on the imported CAD data (.STP) which does not contain model features.

Go to Tabs > Direct Editing (*Available only in Part Environment*)

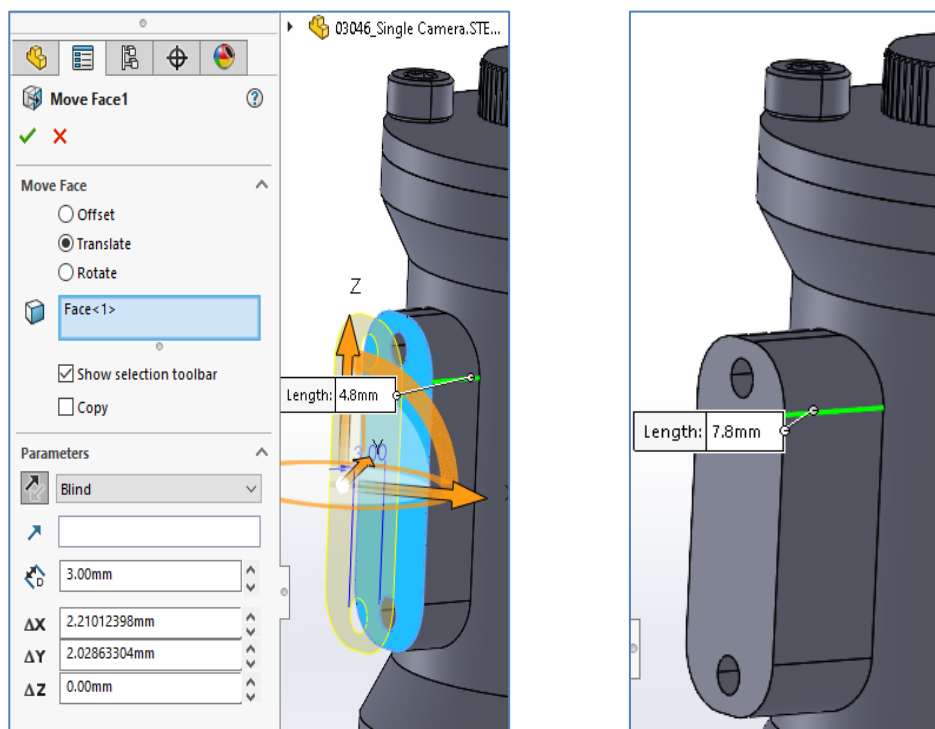
By using these tools, we can modify the geometry as;



- Increasing or decreasing the length, diameter of a body or face
- Adding or removing fillet and chamber
- Split and Combine the Imported Body



Let us look at an example on one of the tools



Move Face

- Open a component from Imported CAD data
- Using the "Move Face", we can lengthen the face from the body by 3mm.
- Click "Move Face" and select "Translate"
- In Face Selection, select the faces which we need to move
- Define the length & direction for the faces to move and click OK.
- Verify the length updated on the model.

*To know more about Direct Editing tools, contact @EGSIndia
Thanks for Reading!!*