SolidTranslate

Onshape – SOLIDWORKS interoperability tool

V 0.22 (alpha)

Created by CADSharp

www.cadsharp.com

For support contact
solidtranslate@cadsharp.com

Contents

Requirements

- SOLIDWORKS 2020 or newer, installed on the same computer as SolidTranslate
    - SOLIDWORKS connected is currently not supported
- Onshape account with the ability to create and modify documents
- Windows 7 or newer

Using SolidTranslate

To launch SolidTranslate, double click the provided .exe file.

When SolidTranslate is launched, the Onshape authentication page will be opened in the default browser. SolidTranslate will act as the authenticated user, and only have access to the enterprise that is chosen.

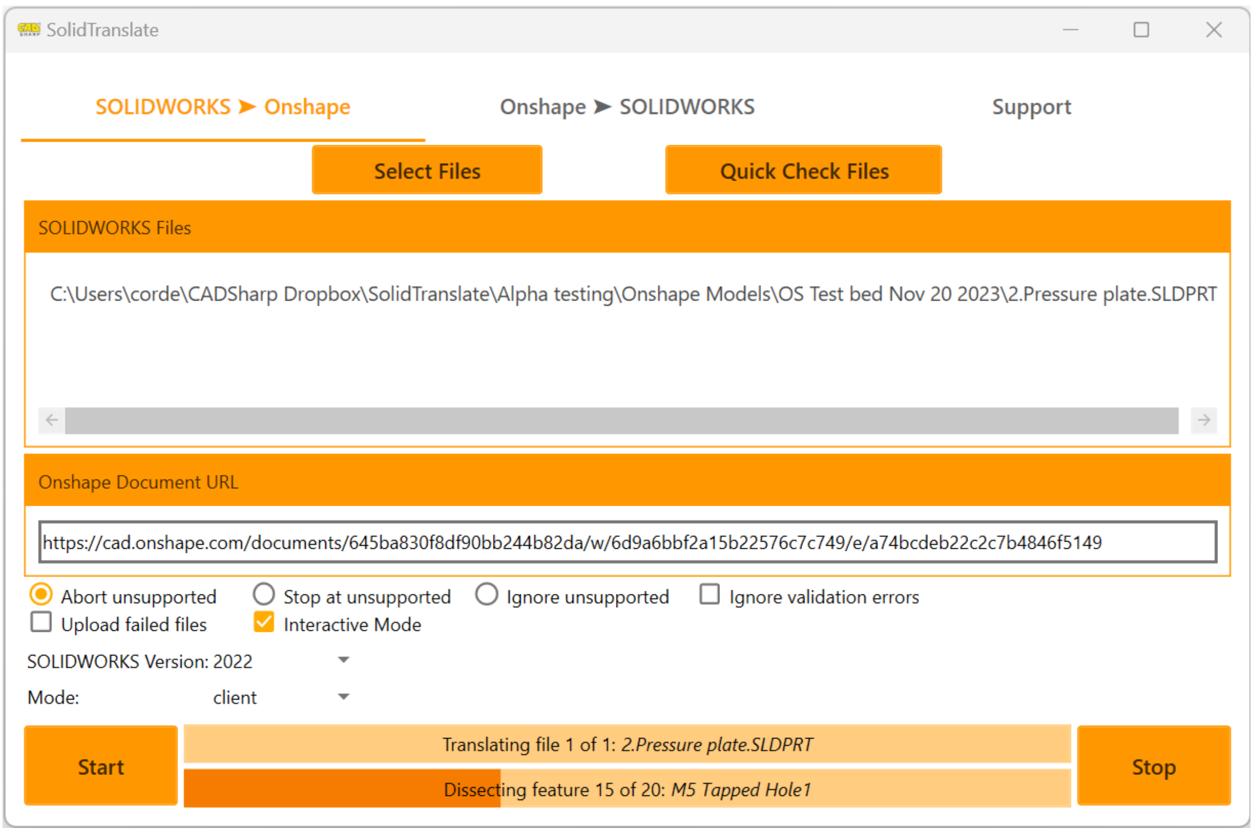

Figure 1: SolidTranslate.exe window

Translation Direction – Select translating from SOLIDWORKS to Onshape or Onshape to SOLIDWORKS.

Select Files – Select files to be translated

Quick Check Files – Files are quickly checked for features of unsupported types. A report is produced on how many files are definitely not supported. Some files may appear supported, but are not due to specific options selected in features. Highly recommended when running on a large number of files.

Onshape Destination URL – SolidTranslate targets an already existing document, new part studios are created in this document, reports are uploaded to this document.

Abort unsupported – When selected, if SolidTranslate encounters an unsupported feature or any kind of issue during translation, the file is completely aborted, and recreation might not occur.

Stop at Unsupported – When selected, if SolidTranslate encounters an unsupported feature, recreation will occur up to that feature.

Ignore unsupported – This option is not recommended for general use, or first time use on a file. If a file is having issues with a specific feature, this option will suppress that feature and all dependent features. Missing features are replaced with a placeholder. This option will result in a model that is different from the source file, but allows the user to potentially fix any issues manually.

Ignore validation errors – This option is not recommended. During recreation, if the file deviates from the source file, these errors will be ignored. This option can be useful if fillets are not recreating as expected.

Upload Failed Files – If a file fails to translate properly, the source file will be uploaded to Onshape. The feature is helpful when sending issues to CADSharp support. Including the Onshape translation, source file, and report are required.

Interactive Mode – If a feature fails to pass validation during translation, translation will be paused and the user will be given a chance to fix the model. Note that this will pause all translations. Do not use interactive mode if you plan only leaving SolidTranslate unattended for long periods of time. To pass validation, users only need to make the translated model match the source model. Users may edit the failed feature or create additional features to pass validation. Once a model passes validation translation continues. Users also have the option to skip validation although this can lead to downstream errors if the model does not exactly match the source.

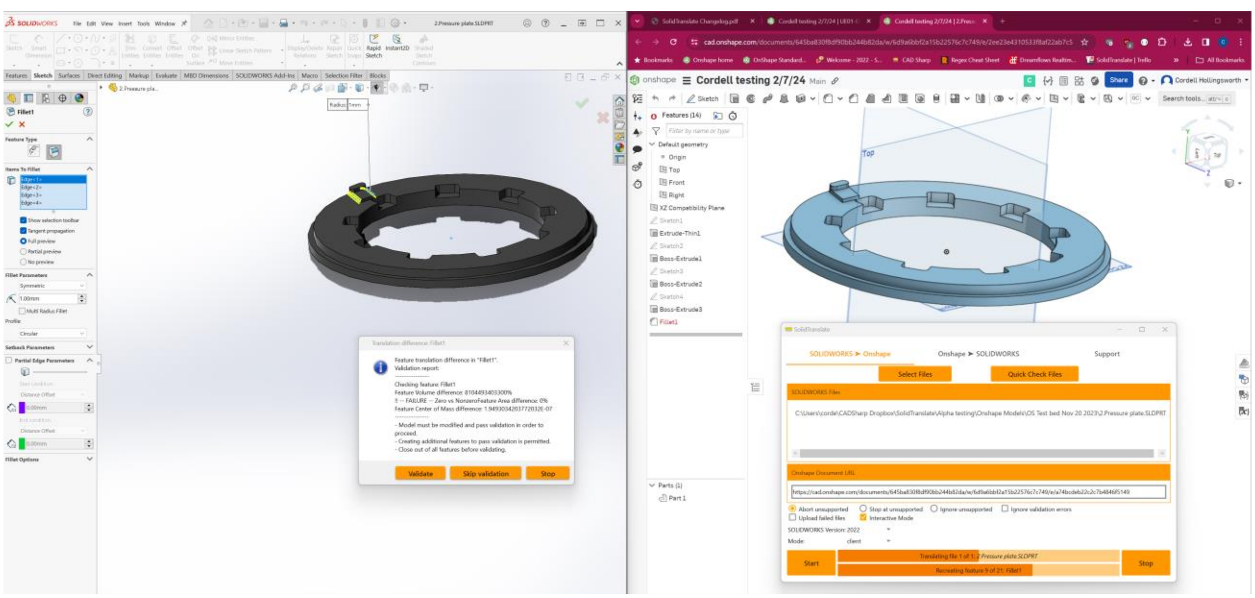

Figure 2: Interactive mode, SolidTranslate pauses translation and waits for the user to fix the translation model so that it matches the source model

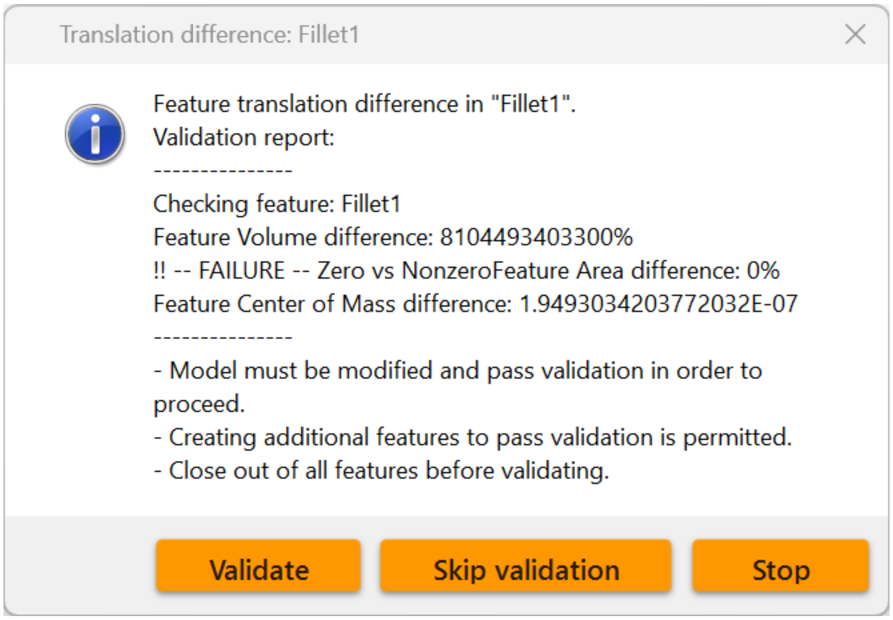

Figure 3: Example of an interactive mode popup message

SOLIDWORKS Version – If multiple versions of SOLIDWORKS are installed on the computer, select the correct version to run SolidTranslate with.

Translation Reports

Translation reports provide the user a method of determining if a model is supported. For each feature in the feature tree, the report will provide information if it was translated successfully, failed, or is unsupported.

Translation reports are uploaded to the destination document in Onshape, and saved locally in the folder containing the translated part file.

Supported Files

Currently only .sldprt files are supported.

List of Support Features

Feature	Requires further development	Unsupported parameters
Sketches	Yes	- Conics - Parabolas - Sketch patterns - Any relation relating to a sketch block

		There are a few corner cases where sketches can fail due to a sequence or combination of constraints, such as multiple offsets in a single sketch. In failed sketches, all sketch constraints are removed. This will affect configurations.
Axis/Coord Systems		
Boolean		
Chamfer		- Vertex reference - Different direction in edges or faces for asymmetrical chamfer
Circular Pattern	yes	- Vary instances - End condition reference - Feature scope
Curve Pattern	yes	- Same as circular pattern - 2^{nd} direction - Define by instance and spacing (only equal spacing is supported_ - Face plane normal orientation - Offset curve
Delete Part		-
Draft		- Step draft - All, inner, outer faces selections - Parting line draft
Extrude	Yes	- Cap end features
Fillet	Partial	- Variable fillets - Asymmetric
Mirror	Yes	-
Pattern	Yes	- Vary instances - End condition reference
Plane		- Any plane tangent constrained to a cone
Revolve		- End condition references unsupported
Shell		
Thicken		- Direction reference
Wrap		
Transform		
Holes	Yes	Advanced hole wizard is not supported - Draft - End condition references - Direction reference - Multiple diameter holes End condition references up to surface needs improvement
Configurations	Yes	Only some simple cases are supported. Design tables are not supported

Equations	yes	Equations that reference a configuration variable are currently not supported
Split		
Curve through points/splines		
Rib		Not all geometry is supported – self intersecting rib shapes will not generate.
Sweep	**Yes**	Only supports solid sweeps. No additional options are allowed in SOLIDWORKS – such as twist correction.

Error logging

Error logs are automatically sent to CADSharp and are associated to the logged in user's email address. Information will include file names, feature names, and error

Support

For support, please contact solidtranslate@cadsharp.com

Please use the "Upload failed files" option and share with editing permission to the support email above.