SolidTranslate Changelog

V 0.22

- Fixed an issue with recreating sketch arcs for OS to SW translations
- Implemented file quick checking for OS dissections
- Implemented reporting for OS to SW translations
- Implemented progress bar for OS to SW translations
- Added support for derived configurations
- Added a time stamp to reports
- Added dissection platform and recreation platform to report

V 0.21

- Fixed an issue with OS to SW translations not starting correctly

V 0.20

- Translation reports
    - Added translation settings to top of report
    - Added a translation failure/success message to top of report, reason for failure displayed with failure message
    - Added variable recreation to translation reports
    - Fixed a bug where translation reports weren't always displayed if translation failed early
    - Fixed a bug where ignored features and ignored feature errors were not added to the end of the report if the report terminated early
    - Features that enter "Interactive Mode" and then have their validation skipped are now added to the list of ignored feature errors
- Fixed corner cases where some unsupported feature modes did not work as intended, unsupported feature modes should now work as follows:
    - **Abort unsupported**
        - When an unsupported feature comes up in PRESCAN, abort translation
        - When an unsupported feature comes up in DISSECTION, abort translation
        - When validation fails after POST-PROCESSING, abort translation
        - When an unexpected error is encountered, abort translation
    - **Stop at unsupported**
        - When an unsupported feature comes up in PRESCAN, dissect up until that feature, recreate up until that feature
        - When an unsupported feature comes up in DISSECTION, skip to recreation, recreate up until that feature
        - When validation fails after POST-PROCESSING, abort translation
        - When an unexpected error is encountered, abort translation
    - **Ignore unsupported**
        - Continue dissection even when an unsupported feature comes up in PRESCAN, recreate passed that feature until failure or end of translation

            - Continue dissection even when an unsupported feature comes up in DISSECTION, recreate until failure or the end of translation
            - When validation fails after POST-PROCESSING, abort translation
            - When an unexpected error is encountered, abort translation
    - **Ignore validation errors:** not an unsupported feature mode, but related
        - When validation fails after POST-PROCESSING**,** continue translation (overrides default behavior)
- UI
    - Fixed a bug where first-time users had no default settings, Unsupported feature options that require one selection were showing up as no selection for these users
    - Added a tooltip to "Quick Check Files" button

- Equations and configurations
    - Refactored equations and configurations translations, more complex equations should now be supported
    - Fixed a bug where dimensions that were identified as global variables and were set as "driven" were translated as "driving"
    - Fixed a number of bugs with adding units to equations
- Added support for midplane planes with non-parallel faces/planes as references
- Fixed a bug where helixes weren't being post-processed
- Fixed a bug where helixes would create a 3D sketch during dissection, that 3D sketch would then dissect as unsupported
- Fixed an issue where nearside/farside countersinks for HoleWizard features were sometimes not being recreated
- Fixed a bug where holeWizards and simpleHoles that used 3D sketches were not dissected as unsupported
- Fixed a bug with continuing dissection after skipping over a library feature
- Fixed a bug with validating translated sketch text
- Fixed a validation error for temporary axes
- Derived configurations now dissect as unsupported
- Reimplemented error logging

V 0.19

- UI Changes
    - Cleaned up UI by removing unnecessary elements
    - Users can now add files to list of selected Solidworks files by double clicking on the list
    - Users can access a context menu to delete all of or a selection of items, by right clicking on the list of selected Solidworks files
    - Editable text boxes can now be escaped with 'Esc' and 'Enter' keys
    - Cleaned up support page
    - Changed SolidTranslate logo
- Translation reports
    - Added more descriptions to messages for unsupported features

  - Implemented reporting for Onshape to SOLIDWORKS translations
- Implemented validation for Onshape to Solidworks translations
- Fixed a bug that caused SolidTranslate to crash when adding a file to a null list of selected files
- Fixed many bug for Onshape to Solidworks translations
- “WeldmentFeature” type feature is now skipped during Solidworks dissection
- Sketches containing infinite length lines now throw unsupported exception

V 0.18

- SW2024 is now supported in SolidTranslate (supported SW versions are SW2020-2024)
- Fixed a bug that caused SolidTranslate to crash before it had even launched when no supported version of SOLIDWORKS was installed
- Implemented flip sided cuts for cut extrudes
- Fixed a dissection issue for sketch regions in boss extrude
- Fixed a dissection issue for extrudes that use the midpoint of an edge as an up to vertex reference
- Fixed a recreation issue for cut extrudes that use the midpoint of an edge as an up to vertex reference
- Fixed a recreation issue for boss extrudes that use the midpoint of an edge as an up to vertex reference
- Now post processing thin extrudes
- Asymmetric fillets now throw unsupported exception (they were previously unsupported but threw no exception)
- Added option to "Interactive mode" to skip validation
- Improved interactive mode pop box message to better explain interactive mode to the user
- Selections errors no longer terminate translation, selections errors that lead to feature errors can now be fixed in “Interactive mode”

V 0.17

- Illegal characters in feature and equation names are replaced
- Fixed support for ignorable external references in some cases for sketches
- Fixes some selection issues
- SolidTranslate will now check for the latest version
- Added interactive mode

V 0.16

- Fixed selection engine issue
- Added merge scope post processing for sweep
- Added additional handling of external references
- Fixed issue with feature names containing #
- Added support for 2nd plane mirror
- Fixed issues with selections
- Fixed issue with split sketch regions
- Fixed issue with configured equations

- Added support for Tapered Pipe Tap holes
- Fixed issue with external references

V 0.15

- Added support for “Flip Direction” in the extrude cut
- Added support for “head clearance” option in hole wizard
- Fixed issues with pattern and mirror feature patterns not merging as expected
- Fixed issue with invalid configuration names

V 0.14

- Support for legacy hole type
- Bug fix in helix
- Updates to UI
- Bug fixes for stopping SolidTranslate
- Improvements to custom hole to include farside countersink, nearside countersink, remove thread option
- Bug fix for feature names with Cyrillic characters
- Support for fractional holes
- Bug fixes for various issues with patterns, skipped patterns, mirroring patterns
- Additional of Onshape to SOLIDWORKS translation
- Addition of “meta” report that details all files in the translation run

V 0.13

- Update revolve to match 173 update
- Fixed issue with hole references missing
- Bug fix for patterning mirrors by feature selection
- Fix sketch mode issue

V 0.12

- Fixed issue with configuration names causing part studios to fail
- Added option to upload failed parts to Onshape to help with future support requests
- Added warnings to ignore unsupported option
- Removed suppression state from unconfigured features
- Updated open contour cut feature to not use construction lines
- Updated planes created pre-2010
- Updated post-processing of linear patterns
- Fixed post processing issue with zero vs non-zero comparison
- Updated planes to use a constraint definition system
- Reports are default to always upload
- Fixed issue with extra delete bodies features appearing in feature tree
- Mate references are now ignored
- Extrude failures in “up to next” are modified to “up to part” in post-processing
- Fixed issue with hole definitions failing to retrieve target in SOLIDWORKS

- Added support for reference axis in constraint planes

V 0.11

- Fixed SOLIDWORKS timeout issue
- Improvements to error logging
- Added placeholder for missing features

V 0.10

- Bug fix for SOLIDWORKS "Toolbar inconsistent" message
- Bug fix for offset cut extrudes
- Bug fix for merge scope in some extrudes
- Added support for helix/wirebody endpoint selection
- Improvements to SOLIDWORKS memory related crashes
- Improvements to handling Onshape internal errors
- Improvements to validation

V 0.9

- Added circular sweeps
- Updated the "Generate Report" button to be a quick check. Features are checked for compatibility only in order to speed up checking if files are going to work. Files that are listed as supported can still fail translation due to unsupported parameters in those features.
- Added skipped Linear, Circular, Curve patterns

V 0.8

- Added basic sweeps
- Added exception handling for Onshape internal errors
- Fixed error messages for design tables
- Fixed error messages for corrupt feature data in SW
- Added helix feature

V 0.7

- Fixes for SOLIDWORKS memory issues
- Fixes for sketch validation
- UI updates to remove debug options
- Ref Axis has been changed from mate connectors over to a custom feature replicating SOLIDWORKS ref axis (using an edge). Fixes constraint related issues.
- Fixed issue where user was able to run multiple SolidTranslate at the same time
- Updates to equations in sketches
- Added additional support for ref planes

V 0.6

- Added support for suppressed configured features
- Added support for equations

- Updated rib to include draft
- Bug fixes

V 0.5

- Bug fixes for selections

V 0.4

- Various bug fixes for selections

V 0.3

- Added support for rib feature
- Added support for tapped holes not supported by Onshape
- Added support for open contour model splits
- Safe sketch mode set to always on
- Various bug fixes

V 0.2

- Added validation to user input URL
- Modified fillet to use “Allow edge overflow”
- Fix for crashed SOLIDWORKS leaving zombie process and locking files
- Fix for user’s having open and unsaved files in SOLIDWORKS
- Added Split feature
- UI updates
- Bug fixes for extrude feature
- Fixed bug with concentric constraints
- Added curve through xyz points feature type
- Added curve through reference points feature type
- Bug fixes for temporary axis in sketches
- Fixed issue with distance dimension attaching to wrong point on arc
- Added exceptions for 3point angles and sketch blocks
- Perpendicular planes
- Temporary axis selection
- Extrude direction fixes
- Post processing for file validation
- Transform scale about centroid
- Added custom dialog box
- Support for pre-2010 planes
- Logging enrichment