

Solidworks STL Exporter Macro

Overview

This is a simple macro written with VBA to automate the STL export process from Solidworks to make the process faster. An STL of the current part is exported to a set folder location.

This macro works with Solidworks 2018, 2019, and 2020

Revisions

Version	Released	Author	Notes
V1.0	03/31/2020	DB	Initial Issue

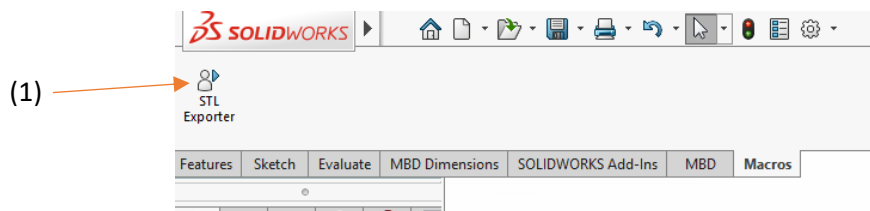
Installation

Download the latest .ZIP folder and unzip.

Assign the macro to the Solidworks tool bar. For help assigning the macro to a tool bar button follow [these](#) steps.

User guide

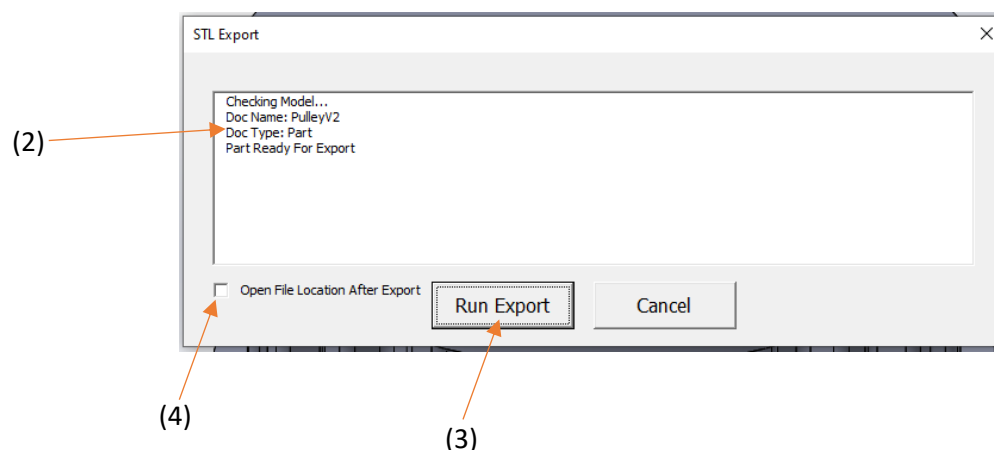
With a part open in the session, run the macro (1).



Document details are checked and displayed in the list-box (2). If the active document is an 'Assembly' or a 'Drawing', the macro will not allow an STL to be exported and the 'Export' button will be disabled. If the document is a 'Part', proceed to the export by clicking 'Export' (3).

Enable the 'Open File Location After Export' check box (4) to open the folder location after export.

This is unchecked by default.



Upon export completion, a message box will appear (5).

All STLs are exported to a folder named 'STL Dump' in the same directory where the macro is saved.

