STL-to-PDMS/E3D™

User's Manual

Version 3.xx

Disclaimer

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1.0 Introduction

1.1 Overall Scope

STL-to-Plant design is a Stereolithographic (.stl) file import for PDMS/E3D. This is a standalone program to transfer geometric data from ASCII and Binary STL files to PDMS/E3D.

STL2PDMS reads polygon mesh data stored in STL files and imports it into PDMS/E3D. A polygon mesh in a STL file is comprised of a set of triangular faces. STL2PDMS recreates the STL model data inside the PDMS/E3D as set of pogo element (Polygon Element).

The sequence of this interface operation is shown diagrammatically in Figure 1-1.
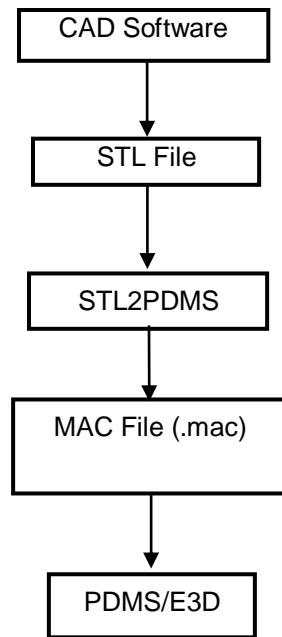
Figure 1-1

1.2 How the Interface works?

1.2.1   The 3D objects created in CAD software such as ProE, Solid Designer, SolidEdge, Solid Works, Unigraphics, Mechanical Desktop, I-DEAS, AUTOCAD, etc., can initially be saved as STL file (*.stl, *.stla,*.stlb).

1.2.2   The windows executable STL2PDMS.exe reads this STL file and generates a macro file (*.mac), which can then be imported into PDMS/E3D.

Note: This interface is compatible with PDMS 11.2 or Later. At this time, this product is fully tested and validated up to PDMS 12 / E3D

1.3 The STL File Format

An STL file is a triangular representation of a 3D surface geometry. STL files contain solids, which are made up of 3-sided facets. A STL file describes a raw unstructured triangulated surface by the unit normal and vertices (ordered by the right-hand rule) of the triangles using three-dimensional Cartesian coordinate system. Most CAD software's are able to output the STL file format, but very few have the ability to import STL files.

STL files may be ASCII or binary data, although binary is far more common due to the resulting size of the CAD data when saved to the ASCII format.

2.0 Installing the Program
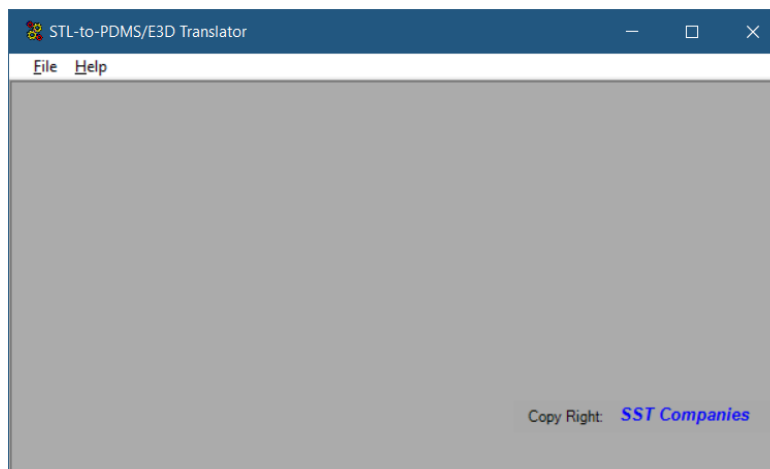
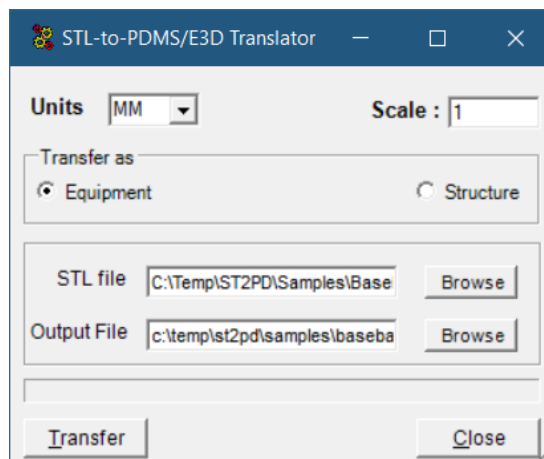Refer Section 1 in SST License Manager User's Manual for details.

3. Limitations

1. Since the components are created as polyhedron, the model is slightly heavier than the usual PDMS/E3D model.

4. Working Procedure

4.1   On launching the program the below form will appear.

4.2   From the menu "File", click "STL-to-PDMS/E3D". A form will be displayed as shown below.

4.3 The STL file does not contain any unit (scale) information; the coordinates are in arbitrary units. In many RP (Rapid product) pre-processing software, the program will try to determine the unit of the part by the magnitude of the dimension. For example if the X/Y/Z size of the part is below 10, it is very likely that it is an inch part. Otherwise it is an mm part.

By default, unit is set to "mm" as shown in the above form. User has an option of specifying the input units as either "mm" or "inches" at this time. If the user want to transfer the STL file other than "Inches" or "mm", then select the input units as "mm" from the "Units" option and specify the corresponding scale factor in the "Scale" text box as shown in figure above. For example, if the user want to transfer in "meters" then specify the Scale as "1000" and selecting the "Units" as "mm".

Note: Scale can also be used for scaling the model to the desired factor.

4.4 Specify the "Transfer As" option as either "Equipment" or "Structure".

    a.   Selecting the "Transfer As" option to "Equipment" transfers the model from STL as Equipment in PDMS/E3D. This option shall be useful in converting the equipment model supplied by vendors in STL format as PDMS/E3D Equipment. User can then add the attributes that are desired for equipment.

    b.   Selecting the "Transfer As" option to "Structure" transfers the model from STL file as Structural Components in PDMS/E3D. The imported model can then be used as a reference model in PDMS, thereby allowing the user to integrate the models developed using different 3D Plant Design software into one system (PDMS/E3D) for detecting the clashes with other disciplines.

        Note: Do not use the "Transfer As" option to "Equipment" for transferring the 3D Plant/portion of a Plant modeled using other Plant Design software as it produce undesirable results in P&ID comparison with 3D, Equipment listing and Bill of Materials.

4.3   Selection of Input / Output File can be done in two ways viz. by entering the name of the File along with the valid path in the text box provided or by navigating through the corresponding button.

4.4   Click the button "Apply" to transfer model to Macro file format. Upon successful transfer, user gets the message "Transfer Successful".

Appendix – A

Instructions on exporting STL files from various CAD packages

The STL file format is produced by almost all CAD software's. For the user the process is often as simple as selecting File, Save As, and STL.

Below are steps for producing high quality STL files from a number of today's leading CAD software. In all cases, export your STL file as a Binary file. This saves on time and file size.

AutoCAD

Your design must be a three-dimensional solid object to output an STL file.

1. Set DISPSILH to 1

2. Set ISOLINES to 0

3. Set FACETRES to 10

4. Relocate the object to the positive X-Y-Z octant

5. Use the STLOUT command to export your STL file

Alibre

1. File

2. Export

3. Save As > STL

4. Enter File Name

5. Save

I-DEAS

1. File > Export > Rapid Prototype File > OK

2. Select the Part to be Prototyped

3. Select Prototype Device > SLA500.dat > OK

4. Set absolute facet deviation to 0.000395

5. Select Binary > OK

IronCAD

1. Part Properties > Rendering

2. Set Facet Surface Smoothing to 150

3. File > Export

4. Choose .STL

Mechanical Desktop

1. Make sure you save your work before attempting this procedure.

2. Set the FACETRES = 6 (As above in AutoCAD instructions)

3. You will have to "Explode" your parametric solid to export an .stl output (*Special note: **DO NOT** save your work after "Exploding")*

4. Use the STLOUT command to export your STL file.

ProE

1. File > Export > Model
2. STL
3. Set chord height to 0. The field will be replaced by minimum acceptable value.
4. Set Angle Control to 1
5. OK

Solid Designer

1. File > External > Save STL
2. Select Binary Mode
3. Select Part
4. Enter filename
5. Enter Max Deviation Distance of .01mm
6. Click OK

Solid Edge

1. File > Save As
2. Set Save As Type to STL
3. Options
4. Set Conversion Tolerance to 0.001in. Or .0254mm.
5. Set Surface Plane Angle to 45.00º
6. Save

Solid Works

1. Tools > Options > Export
2. STL Options
3. Set Quality to Fine
4. File > Save As
5. Set Save As Type to STL
6. Save

Unigraphics

1. File > Export > Rapid Prototyping
2. Set Output type to Binary
3. Set Triangle Tolerance to 0.0025
4. Set Adjacency Tolerance to 0.12
5. Set Auto Normal Gen to On
6. Set Normal Display to Off

CADKEY

1. Go to the File menu, then Export, then STL
2. Type in a filename and select OK.

3. A new window will appear with options for the export

4. Change Format to Binary

5. Change Facet Tolerance if a higher resolution STL file is desired

Additional export tolerance options are in the Solids99 Configuration window accessed from the Tolerance section of Configuration.

Set Triangle Display to On