



Tutorial

Geometry Import, Mid-surfacing and Mesh Generation

Updated on: March 3rd, 2025

Tested with: SDC Verifier 2024 R2

- This tutorial demonstrates how to rapidly transform 3D geometry into a 2D mesh using SDC Verifier.
- Learn how to import Geometry file into SDC Verifier;
- Learn Mid-surfacing techniques;
- Learn a quick and effective technique for 2D mesh creation;
- Understand the key steps and settings for optimal results;
- Improve your FEA preparation efficiency

Import Geometry

1

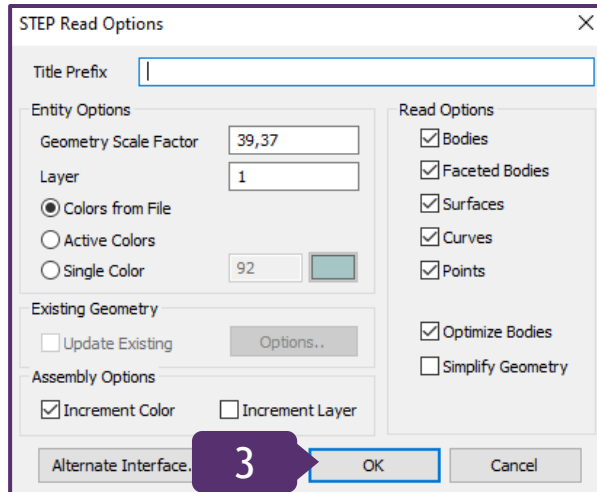
In File section of the Ribbon, press *Import*; Select *Geometry*

2

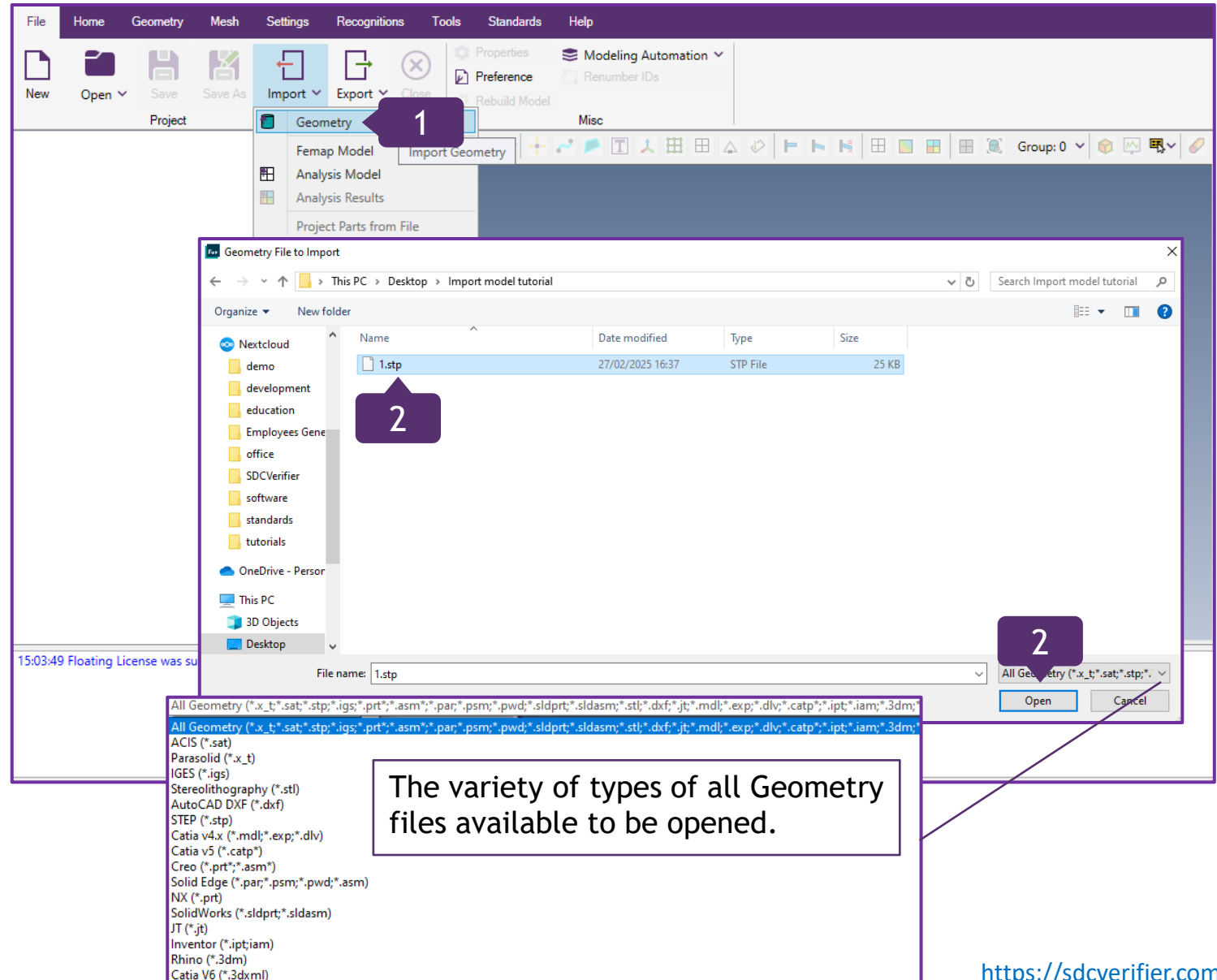
Select the Geometry file and press *Open*

3

Press *OK*



Import function creates a new SDC Verifier project from current model or updates opened project with imported CAD geometry, finite elements model or analysis results.



SDC Verifier automatically creates Parasolid geometry from the data in the STEP file. Read Options regulate how the translations of specific STEP entities are controlled.

Is used to scale the units of the model. This can be extremely useful with some geometry packages that default to a standard for output units (such as meters), even though other units to model the part might have been used.

A layer of the solid model can be defined.

It is possible to read ACIS Assembly files and to convert them to Parasolid as well. There is a choice to have each solid from the assembly on a different layer, as well as a different color. When OK is selected, SDC Verifier will open the ACIS-to-Parasolid converter and read the file. No other action is required.

STEP Read Options

Title Prefix

Entity Options

Geometry Scale Factor: 39,37

Layer: 1

Colors from File (selected), Active Colors, Single Color: 92

Existing Geometry

Update Existing (unchecked), Options...

Assembly Options

Increment Color (checked), Increment Layer (unchecked)

Read Options

Bodies (checked), Faceted Bodies (checked), Surfaces (checked), Curves (checked), Points (checked), Optimize Bodies (checked), Simplify Geometry (unchecked)

Alternate Interface..., OK, Cancel

Bodies - when ON, imports manifold solid B-Rep entities;

Faceted Bodies - when on, imports faceted B-Rep solid entities (i.e., often found in a JT file);

Surfaces - when on, imports shell-based surfaces and surfaces without topology;

Curves - when on, imports wireframe curves;

Points - when on, imports wireframe points;

Optimize Bodies - when on, attempts to heal edges, remove redundant topology, and share geometry;

Simplify Geometry - when on, attempts to recover the analytic definitions for the B-spline geometry in the part (off by default).

Mid-surface the Geometry

1 In Geometry section, expand Midsurface;
Select *Automatic...*

2 Expand Method;
Select on *Solid*

A mid-surface is a 2D representation of a 3D structure,
positioned at the geometric center of the original solid.

The screenshot displays the SDC Verifier software interface. The 'Geometry' tab is active, and the 'Midsurface' dropdown menu is open, with the 'Automatic...' option selected (indicated by a purple circle with the number 1). Below this, the 'Entity Selection - Select Surfaces to Midsurface' dialog box is open. In this dialog, the 'Add' radio button is selected, and the 'Method' dropdown is open, showing the 'on Solid' option selected (indicated by a purple circle with the number 2). A green callout box on the left explains that the 'Automatic...' option uses face pairing technology within the Parasolid modeling engine to generate a mid-surface representation. The main 3D view shows a yellow, hollow, stepped rectangular part with its mid-surface highlighted in green. A coordinate system (X, Y, Z) is visible in the bottom left of the 3D view.

Entity Selection - Select Surfaces to Midsurface

☒ Add ☐ Remove ☐ Exclude

ID to by

or Group

Select All

- ☒ ID
- ☐ Color
- ☐ Layer
- ☐ Property
- ☐ in Region...
- ☐ From Node...
- ☐ From Element...
- ☐ using Curve
- ☒ on Solid

Automatic... option attempts to employ face pairing technology within the Parasolid modeling engine to automatically generate a mid-surface representation for a solid part or between chosen surfaces.

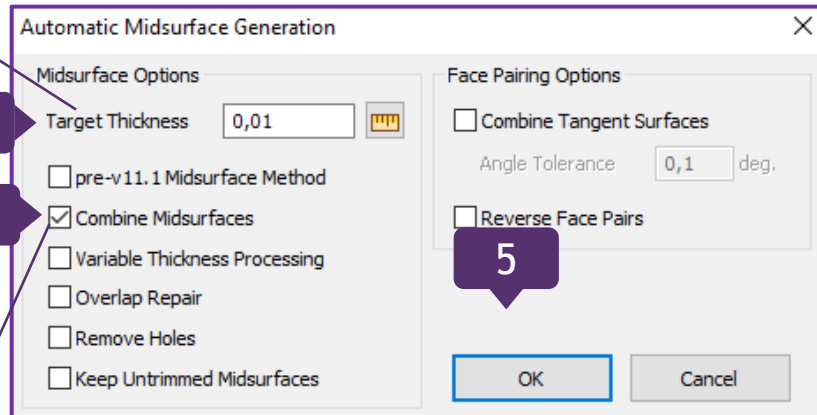
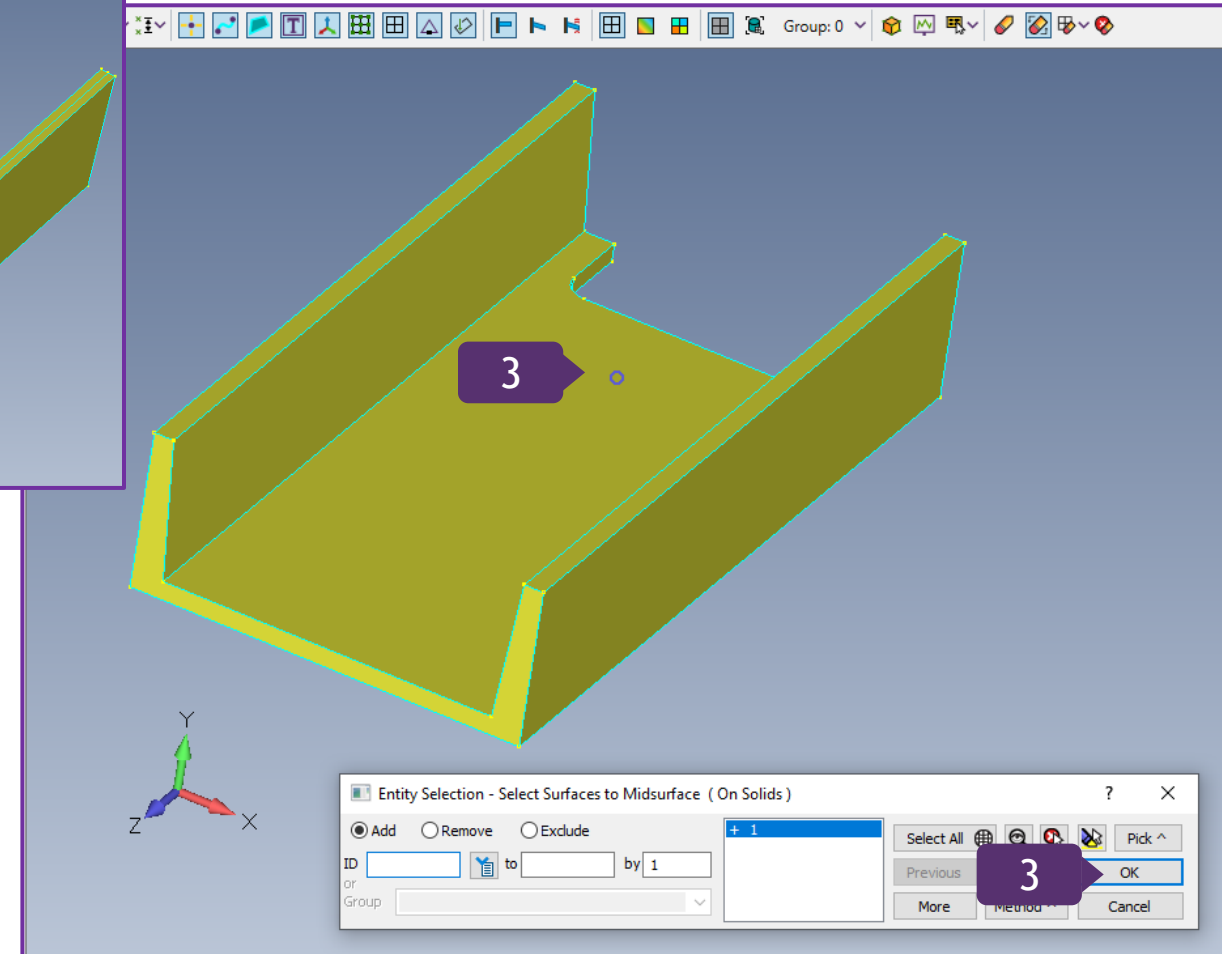
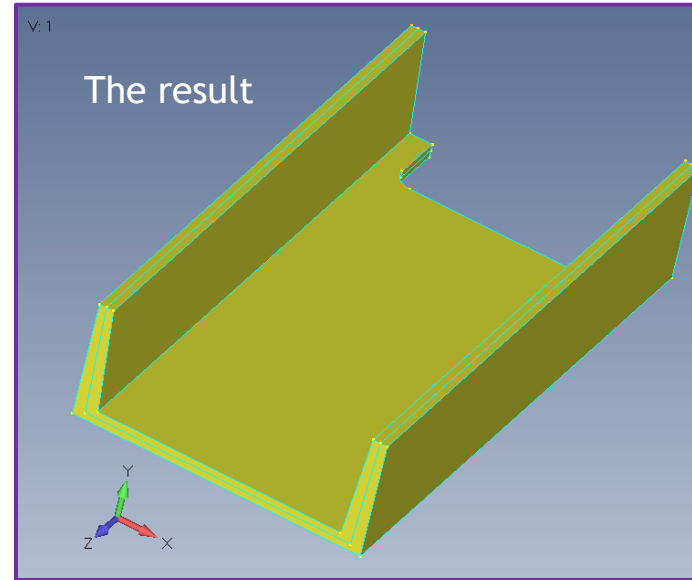
Mid-surface the Geometry (Continuation)

3 Select the Solid and press **OK**

4 Target Thickness: **0,01**;
Combine Mid-surfaces: **ON**

5 Press **OK**

Target Thickness equals mid-surface tolerance. Any surface with a distance, equal or less than the Target Thickness will have a mid-surface generated.



Combine Mid-surfaces option is enabled in case it is needed to combine multiple solids in a single command.

1

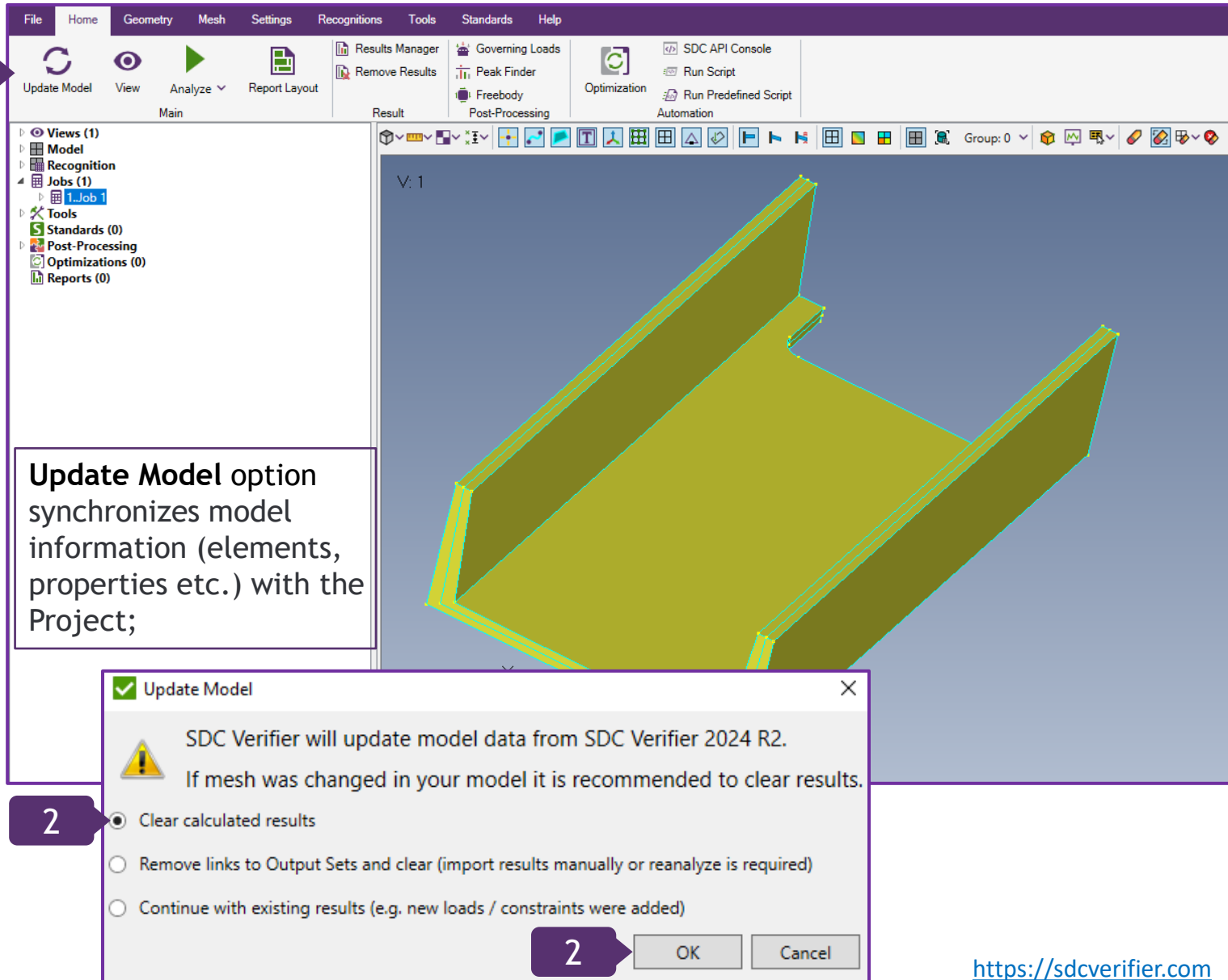
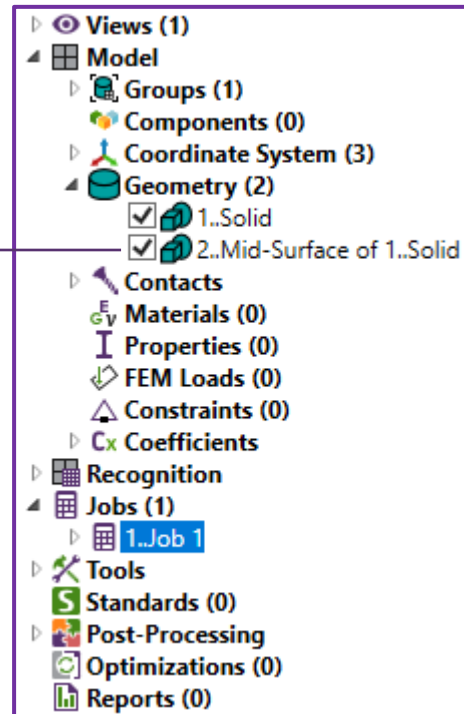
In Home section, press *Update Model*

1

2

Select *Clear calculated results* and press OK

The mid-surface of 1..Solid has been created



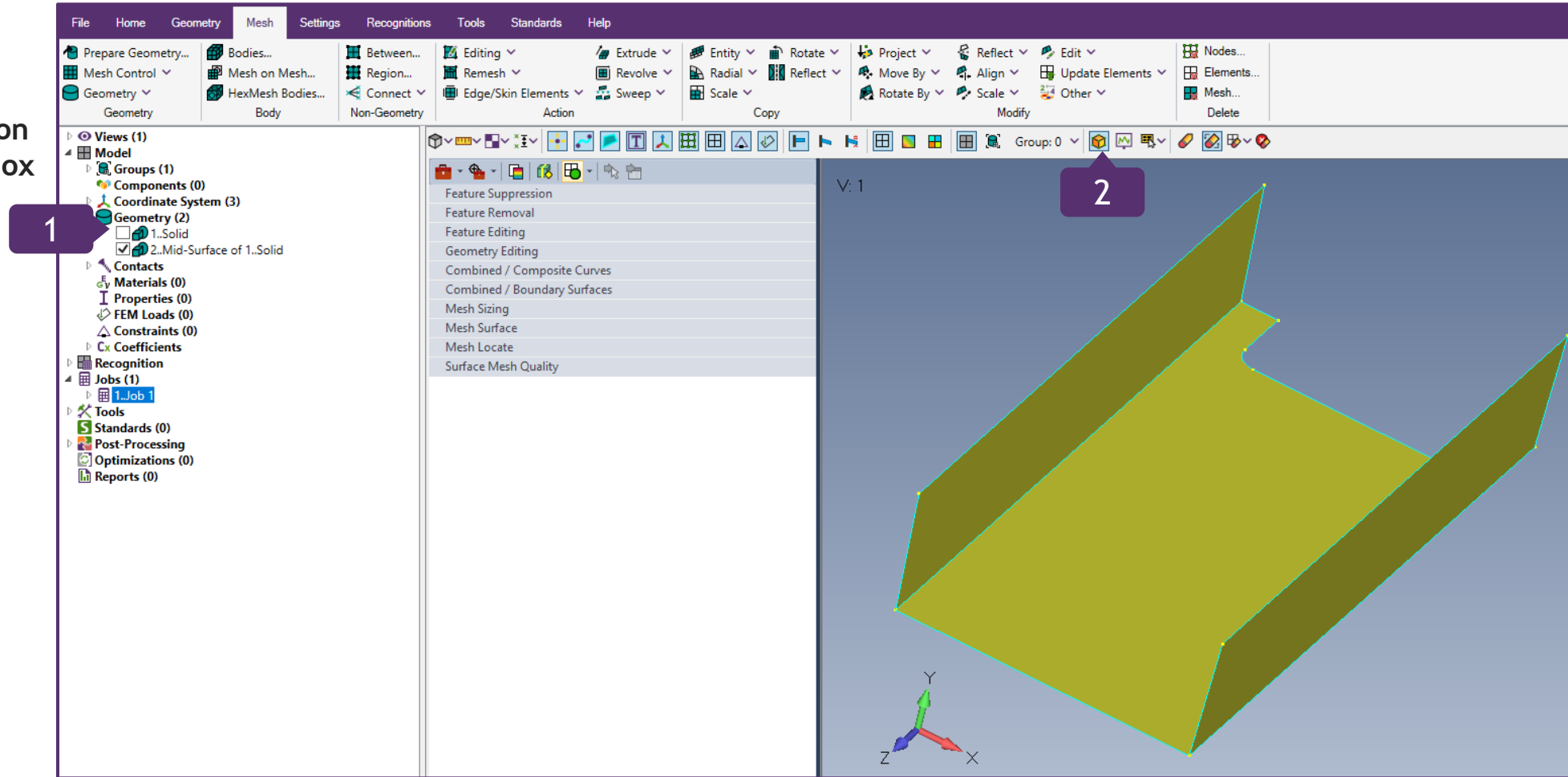
Switch Meshing Toolbox On

1

Inactivate 1..Solid

2

Activate Meshing button to open Meshing Toolbox



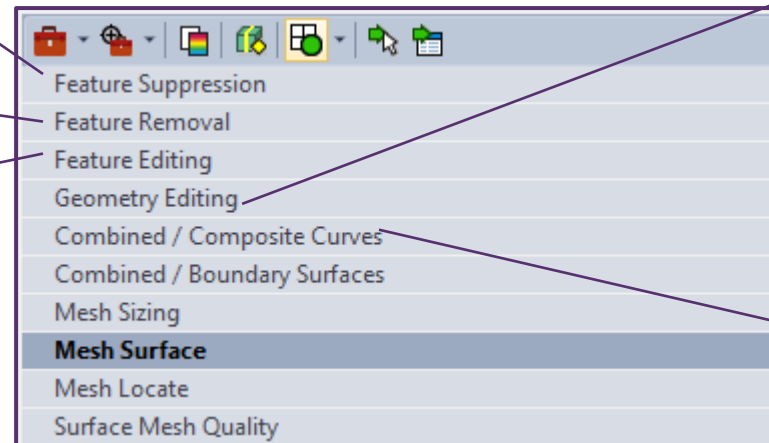
The **Meshing Toolbox** contains individual tools which can be very helpful during the meshing process. There are tools which allow to simplify geometry; modify geometry to create a better mesh; create “combined” geometric entities for meshing purposes using several underlying geometric entities; change the mesh size, biasing, and other options on any number of curves interactively; move any number of nodes dynamically while seeing the mesh update; and plot the element quality in the graphics window.

This Tool suppresses loops (curves of internal holes on surfaces and solids, “base curves” of bosses and extrusions on solids), curves (usually relatively small in size), and surfaces (usually sliver surfaces, not fillets or chamfers). Suppressed geometry still exists in the model and can be restored at any time.

This tool is used to split or otherwise modify curves or surfaces to create geometry for the purpose of producing a mesh with better quality elements.

The Tool is used to permanently remove geometric entities to simplify geometry.

Used to make relatively basic alterations to Geometric features. Examples include modifying the position of a hole, boss, or rib, changing the length/width of an extrusion or entire part, revolving a face to create an angled wall or extending a revolved body, etc.

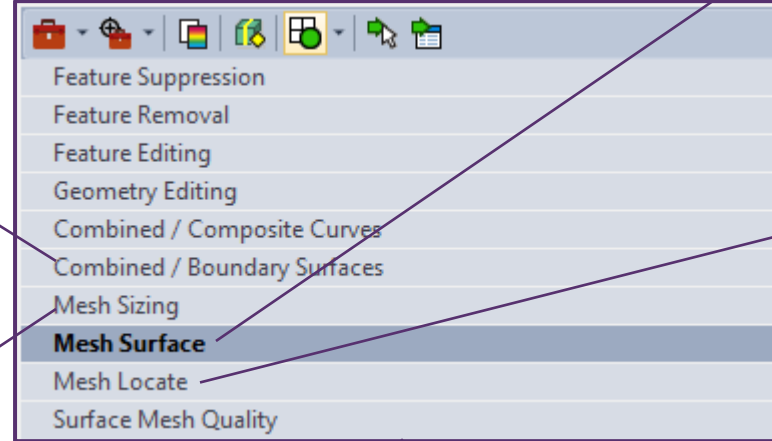


In some cases, combining several smaller curves along the edge of a surface will allow you to create a higher quality mesh on the surface. This tool allows you to combine curves by choosing the curves themselves or a point that two curves share. A “Composite Curve” will be created in SDC Verifier, which will be used for mesh sizing purposes instead of the underlying curves. There are also options for splitting a “composite curve” at a selected point or removing any of the underlying curves.

This can be especially helpful when there are “sliver surfaces” next to a much larger surface. By combining the selected surfaces into one boundary surface, all of the internal curves can be ignored during the meshing process. Boundary surfaces can be created by selecting a curve shared by multiple surfaces or choosing the surfaces themselves. Also, any underlying surface can be removed from a boundary surface or split along a chosen curve.

Combines the options used to set mesh sizing and node spacing on curves (Mesh → Mesh Control → Size on Curve) with the “Add, Subtract, and Set To” functionality of the Mesh → Mesh Control → Interactive command.

Allows surfaces to be meshed or re-meshed, using various surface meshing methods, options, and/or mesh approaches. Set or change options for Meshing Attributes, Property, Mesh Sizing, Element Shape, Meshing Method (Free or Mapped), Smoothing, Offsets, Quad/Tri Layers, and others. Choose Auto Mapped Approach or define approaches manually to quickly converge on a higher quality mesh.



This tool allows to make small changes to an existing mesh simply by moving one or several nodes without changing the number of elements, while making sure that as moving the node or nodes dynamically, they remain attached to specified solid(s), surface(s), and curve(s), or if you have no geometry, follow the overall topology of the selected standalone mesh. There are also options to move the selected nodes by a defined amount, continually smooth the mesh as the nodes are moved, and allow the moved nodes to no longer be attached to surfaces or curves.

When the Mesh Quality Toggle in the Meshing Toolbox is set to “ON”, this tool allows you to graphically see an element quality value plotted on each element similar to a contour/criteria plot. There are several different element quality types which can be selected and each type has default automatic values, but user-defined values can also be specified. Additionally, the minimum and maximum quality values for the specified “quality type” are listed in the bottom fields of the tool.

1 Expand *Mesh Surface*

2 Property: *Use Meshing Attributes*;
Mesh Sizing: *Size All, Connect*

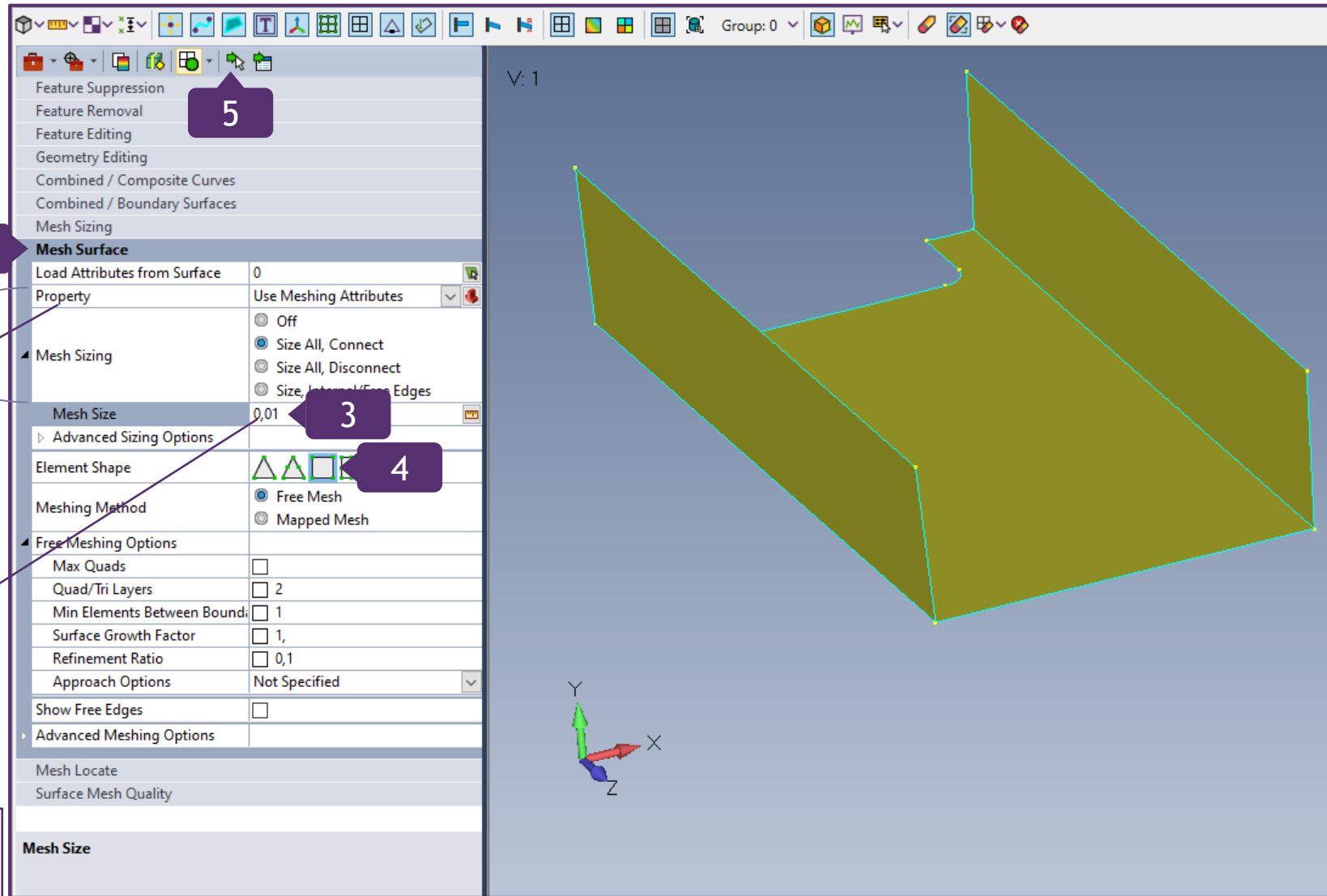
3 Mesh Size: *0,01*

4 Element Shape: *Rectangular*

5 Press *Select* button

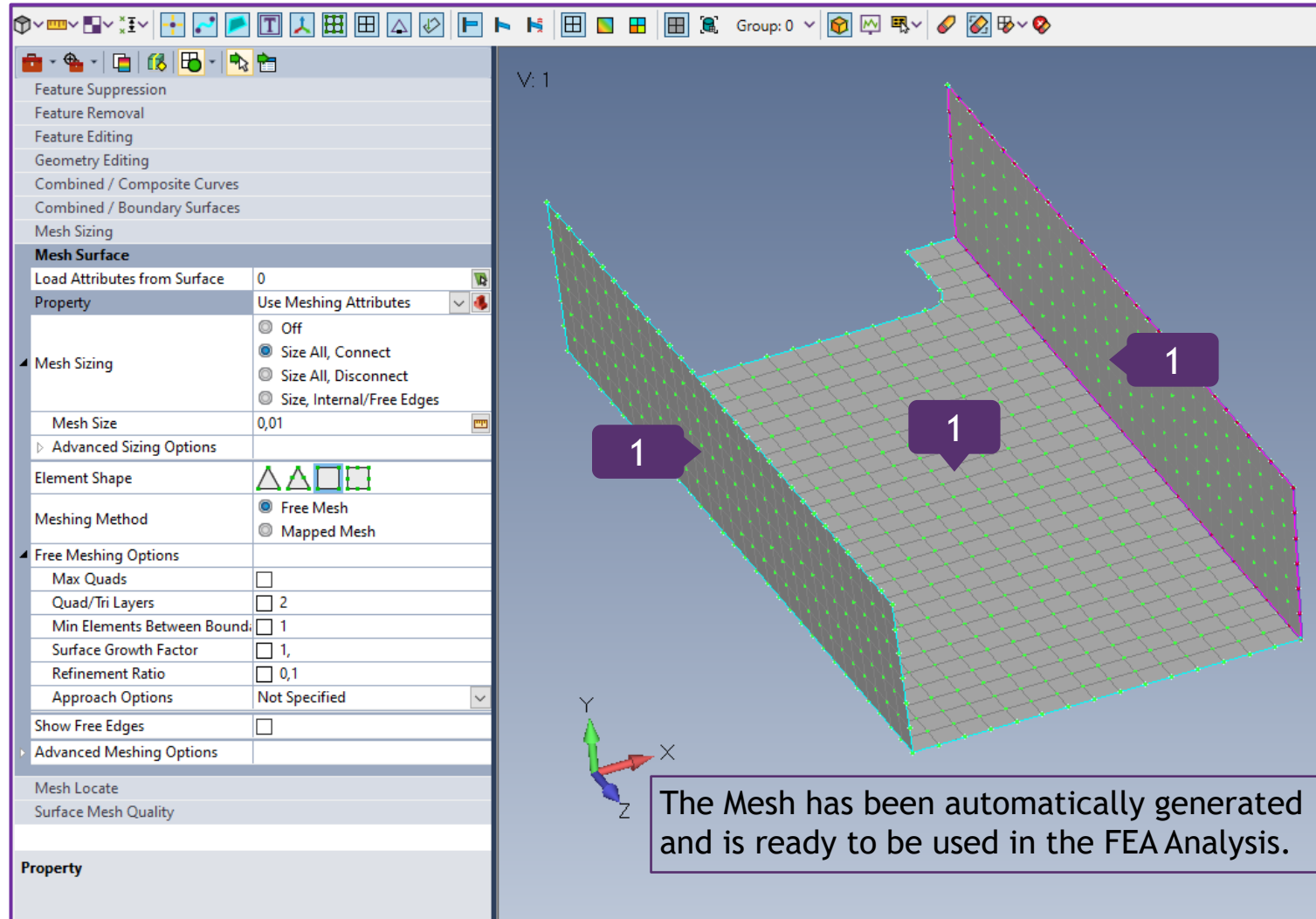
Property allows to choose an existing property in the model to mesh/re-mesh surfaces. The default value is to “Use Meshing Attributes” which have been applied to the surface from previous meshing commands. A new property can also be created.

The Mesh Size value may be entered directly or can be calculated by clicking the “Measure” button and selecting any number of surfaces.



1

With left-clicking, select all Surfaces



Save the Project

1

In File section, press *Save As*

2

Select the proper place for the model to be saved and give it a relevant title

3

Press *Save*

