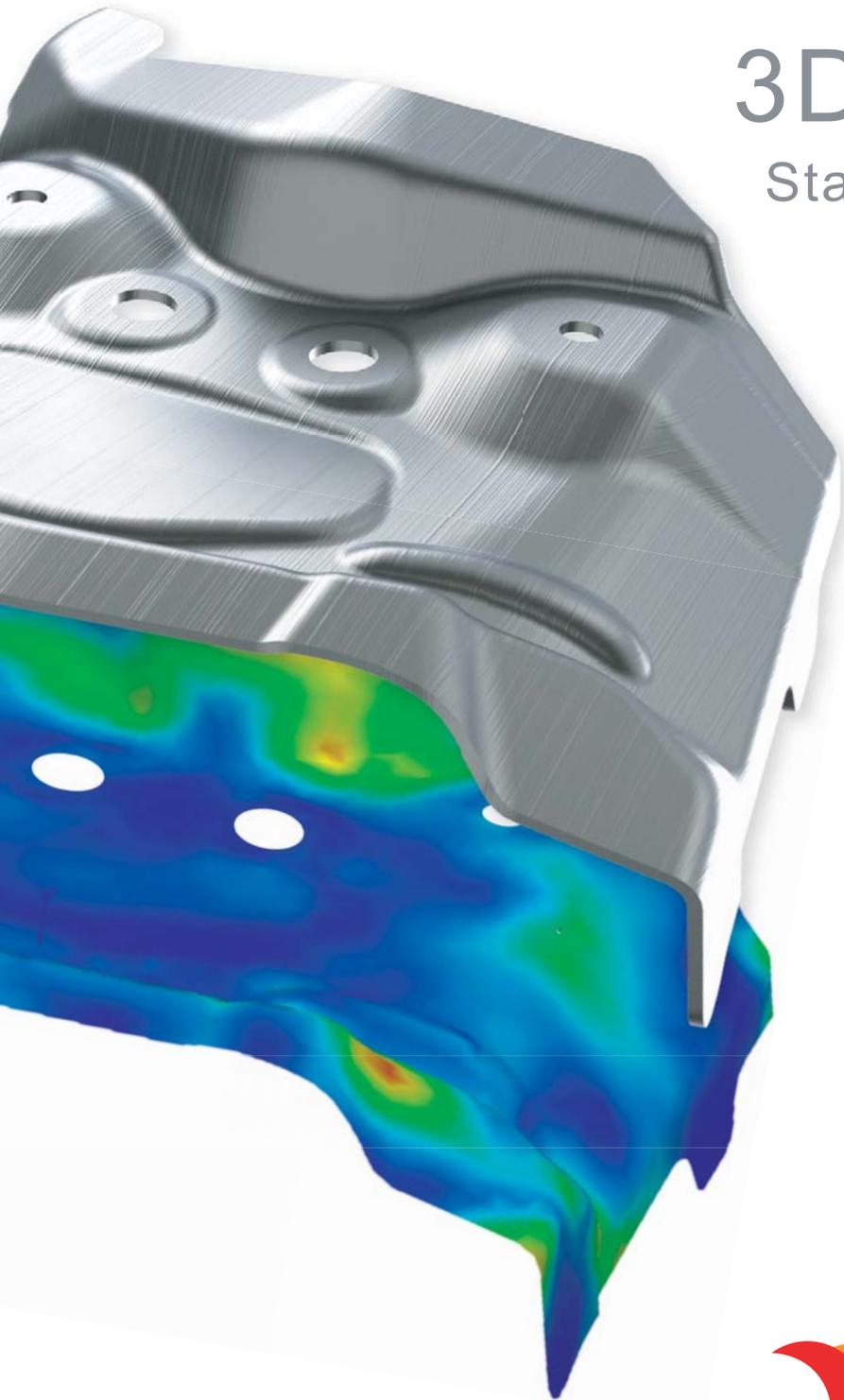


# 3D QuickTools Limited Training Manual

## 3DQuickForm

Standard & Professional



powered by



## Start a new project

3DQuickForm support multiple projects and you can compare the results easily.

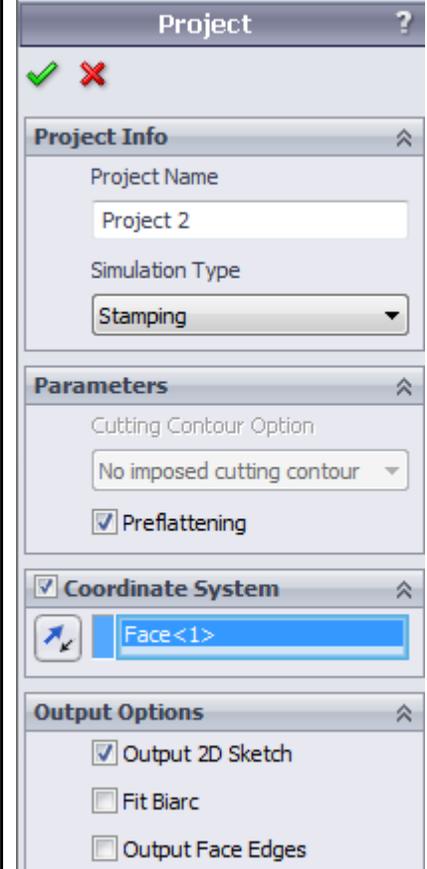
To start a new project:

1. Click 3DQuickForm, New Project Or Right click on Manager tree



2. From the properties manager, accept all the default Parameters. Select the co-ordinate system representing the punch direction in Z-
3. Click OK when finished.

## Project Properties Manager

	<p>Project Name – Input project name Simulation Type – Stamping &amp; Flanging</p> <p>Parameters Preflattening – Turn ON if the part is having undercut area</p> <p>Coordinate System – Punch direction can be defined by coordinate system; Reference plane or planar face</p> <p>Output Options Output 2D Sketch – Turn OFF for Flanging and Initial blank simulation have 3D output. Check this box to get a 2D sketch. Fit Biarc – Fit 2D output by line or arc based on the tolerance. Output Face Edges – Face edges will be mapped to the result sketch.</p>
--	---

## Part Definition

To define a part for analysis:

1. Click 3DQuickForm, Project, Set Punch Part or Right click on Project folder
2. Select the faces representing the part
3. Input material thickness
4. Click OK when finished

## Apply Boundary Conditions

Boundary conditions can be assigned to the part.

Boundary conditions		Availability
Fixed		Standard 3DQuickForm
Symmetry Plane		Standard 3DQuickForm

## Mesh the part

To mesh the metal part:

1. Click 3DQuickForm, Create Mesh
2. Click OK to accept the default mesh size
3. If no mesh is created, repeat step 1 and half both global mesh size and minimum mesh size.

Note: Finite element analysis is very sensitive to mesh quality, visual check the mesh output and make certain correction on the model is required sometimes.

### Show Unmeshed Faces

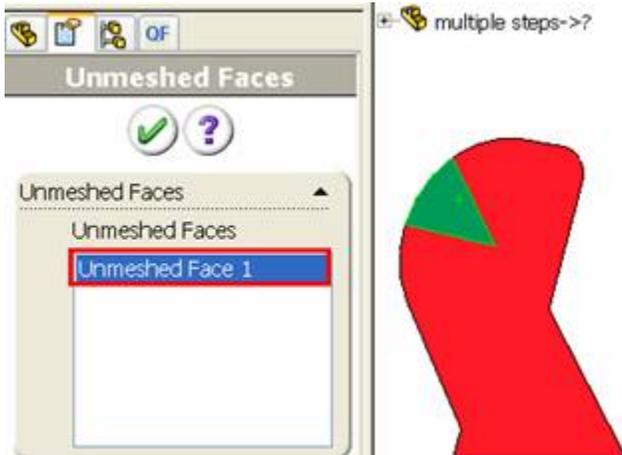
Face cannot be meshed in some occasions due to improper mesh size.  
To identify unmeshed face when the warning message appears



1. Click OK and RMS Project folder, select **Show Unmeshed Faces ...**



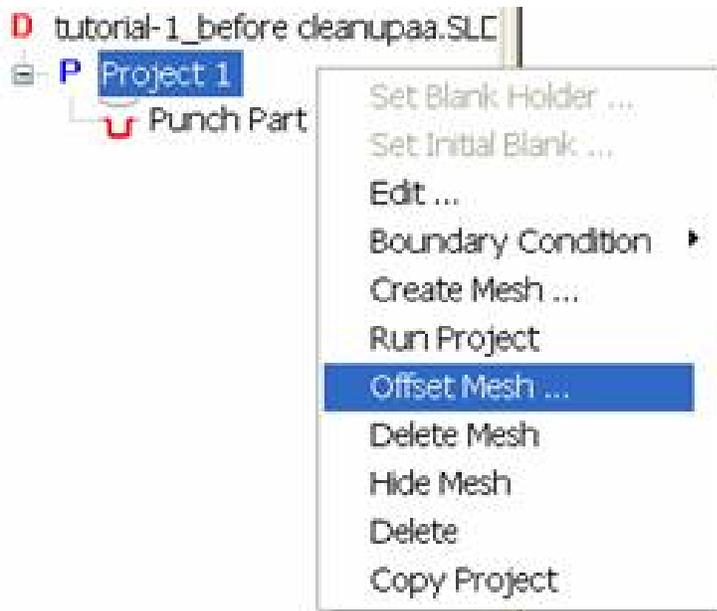
2. Select listed face and 3DQuickForm will zoom the unmeshed face

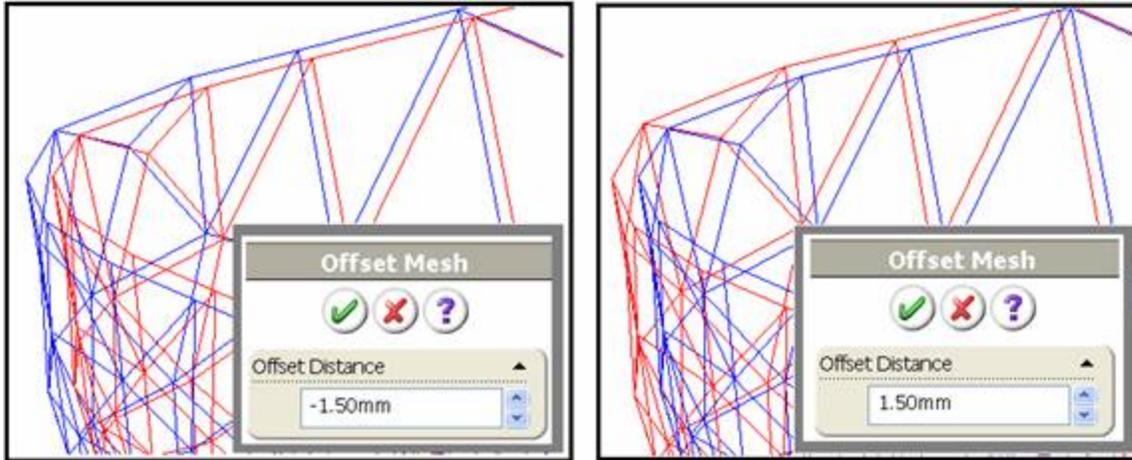


3. Re-mesh the project by using preserved edge or reduce the mesh size.

## Offset Mesh

Default mesh created can be offset, the sign of the offset determine the offset direction, the preview of the offset mesh is **RED** in color





## Set Preserved Edges

This setting can preserve the geometry accuracy after meshing

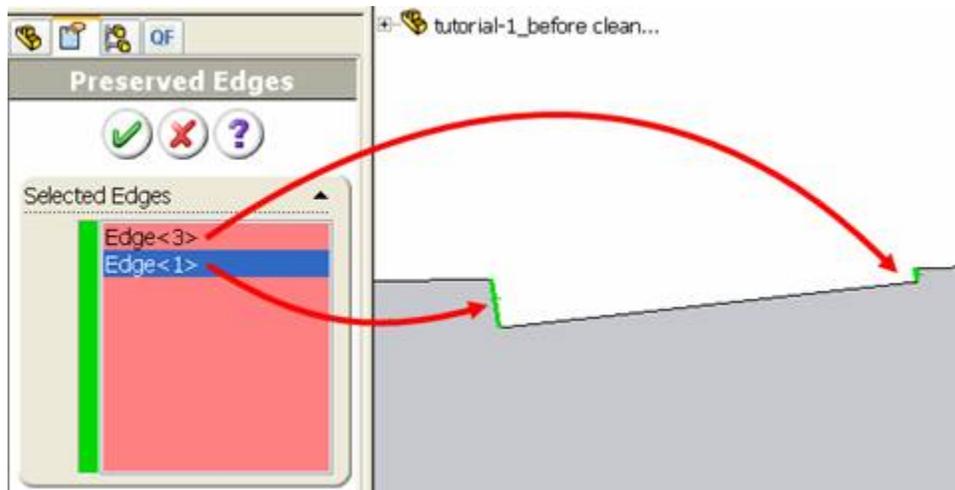
To preserve edges tolerance:



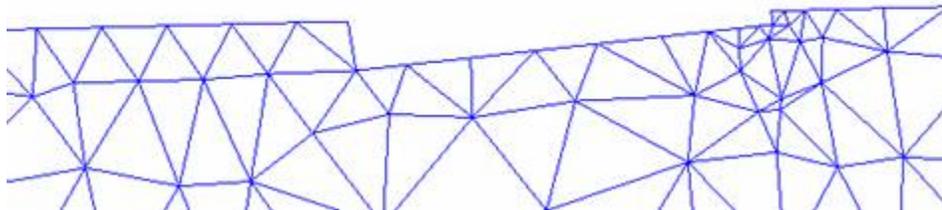
1. RMS **Punch Part** folder and select **Set Preserved Edges**



2. Select Edges which require accuracy to be preserved



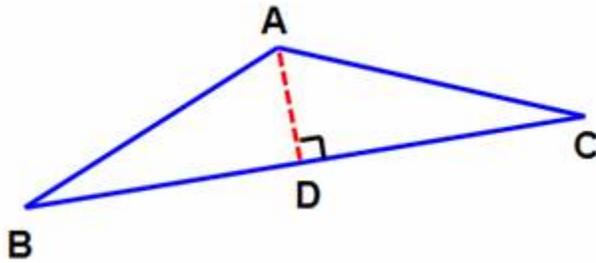
3. Click OK and re-mesh the project. The selected edges are preserved.



## Mesh Consideration

### Aspect Ratio

3DQuickForm generate triangular mesh. In order to have successful simulation, the mesh quality must be good enough. Aspect ratio is used to measure the quality of mesh which is defined as the longest side divided by min altitude.



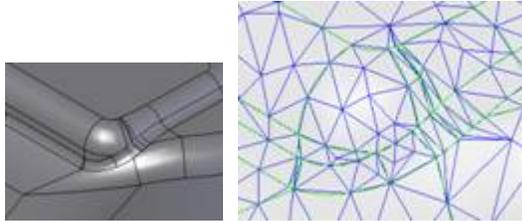
$$\text{Aspect Ratio} = BC/AD$$

The best aspect ratio is the case of equilateral triangle. The worst aspect ratio the solver can tolerate is less than 10.

As the mesh quality is driven by the faces from the punch part, modification of the part is required sometimes in order to have a more stable simulation.

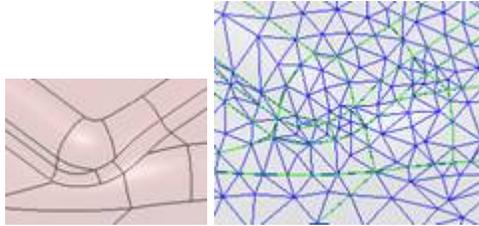
## Example

Meshes from tiny faces usually create poor aspect ratio mesh



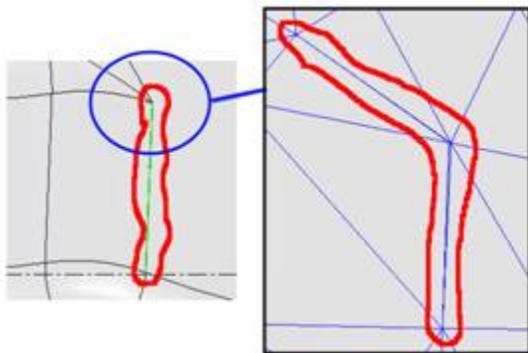
Element can not be greater than the face size, the aspect ratio of those elements from tiny faces usually bad.

Solution



Use SolidWorks Delete Face and fill with tangency option to simplify the region to get a better mesh

**Long needle like face create VERY BAD elements which can affect the accuracy of the result up to 5%**



Solution

Use delete face and fill with tangency option

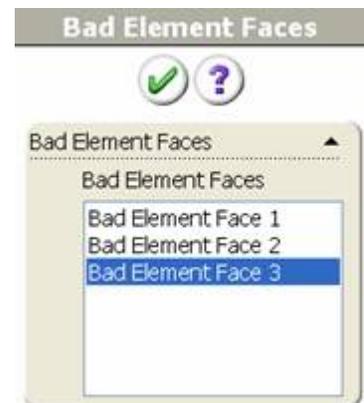
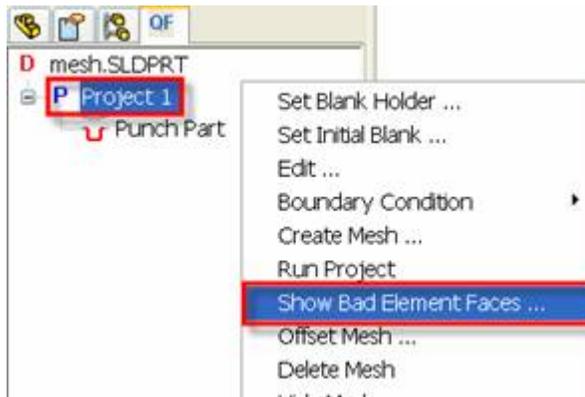


## Show Bad Element Faces

If bad element exists (Aspect ratio > 10), a command is available from Project folder shortcut menu to identify the faces generate the bad quality element.

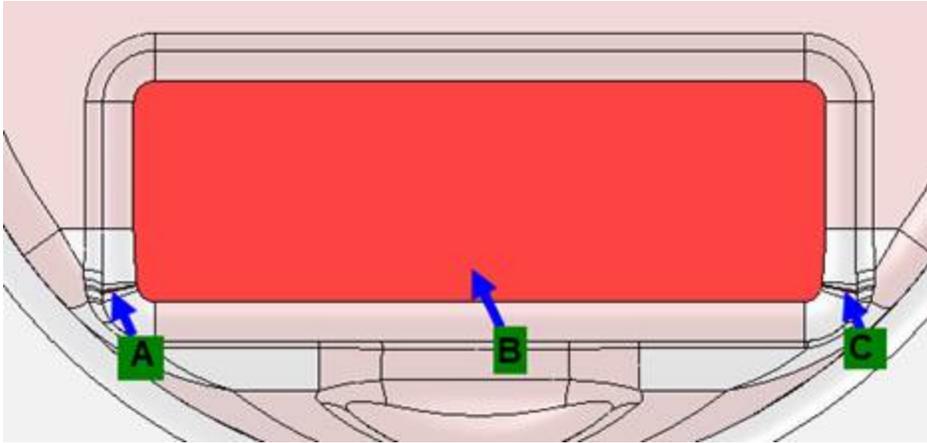
To identify bad element faces:

1. RMS the project tree, if **Show Bad Element Faces** is available, select it



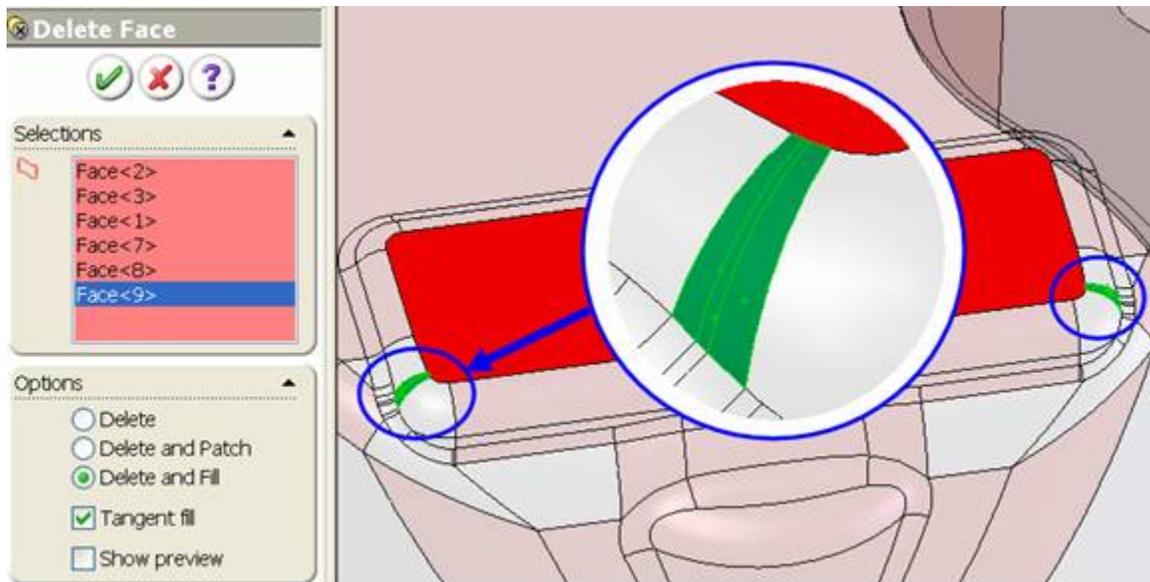
2. Select the face from the list and 3DQuickForm zoom it to fit.
3. Click OK to exit

Normally, bad faces are created from tiny faces and the face adjacent to the tiny face will be affected as well. The following 3 faces are reported by **Show Bad Element Faces**. And only Face A & B have to handle.



To get rid of the bad elements:

1. Click Insert, Surface, Delete Surface
2. Select the tiny faces and it adjacent faces



3. Click Delete and Fill, check Tangent fill
4. Click OK to finish
5. Re-mesh the project.

Note: Both meshes output were accepted by the Solver and the result only have 1% deviation.

## Run the Project

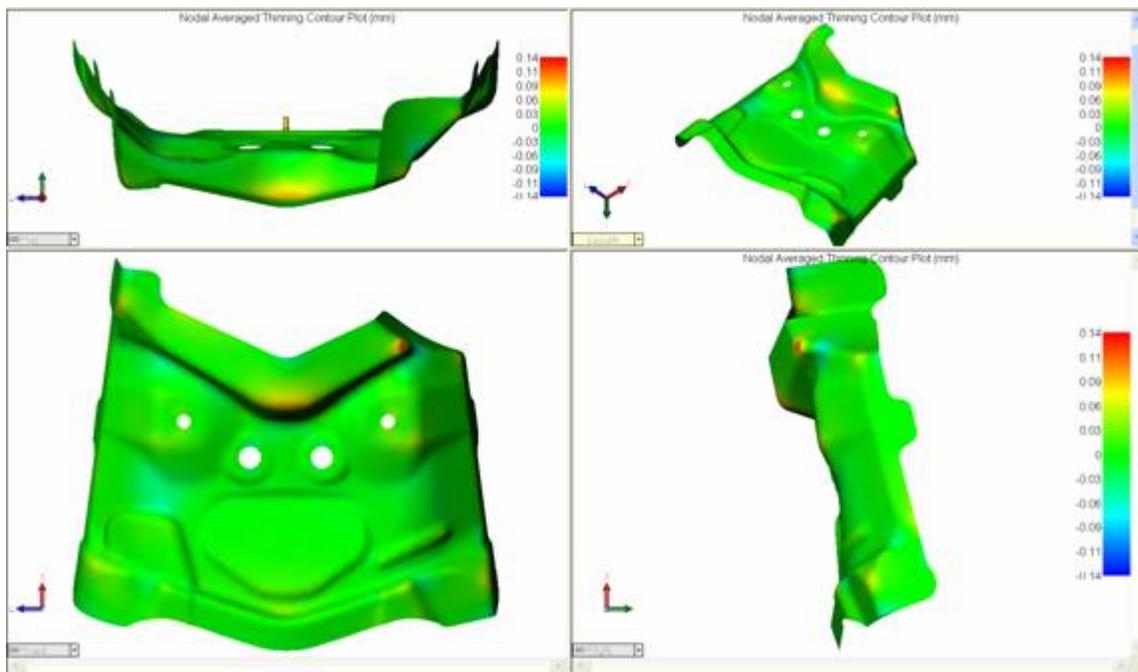
To start the analysis:

1. Click 3DQuickForm, Run Project to analysis the part
2. The blank sketch will be created, the name for the sketch follows the project name with serial number appended.

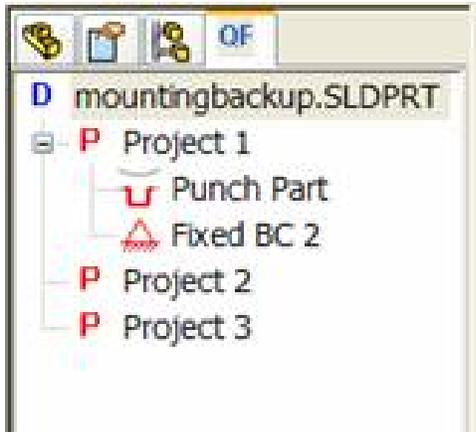
Note: In case the simulation is not converged, refine the mesh.

## View the result

Click 3DQuickForm, Show thinning to show the thinning. **Blue** is thicker than blank thickness, indicating wrinkling, **RED** is thinner than blank thickness indicating higher chance of fracture.



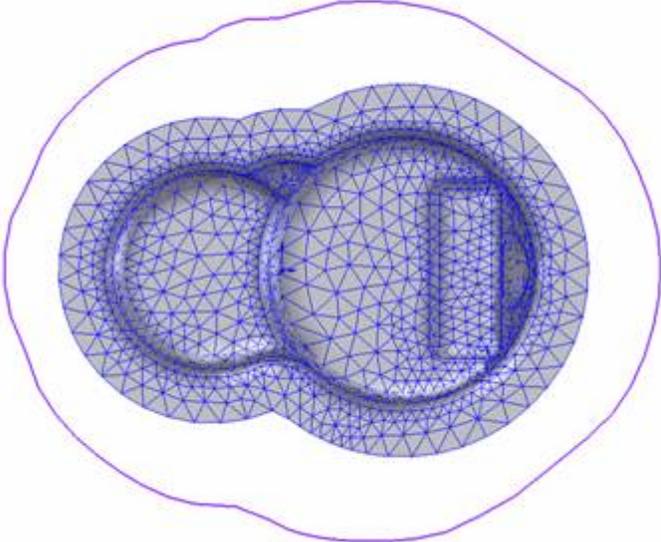
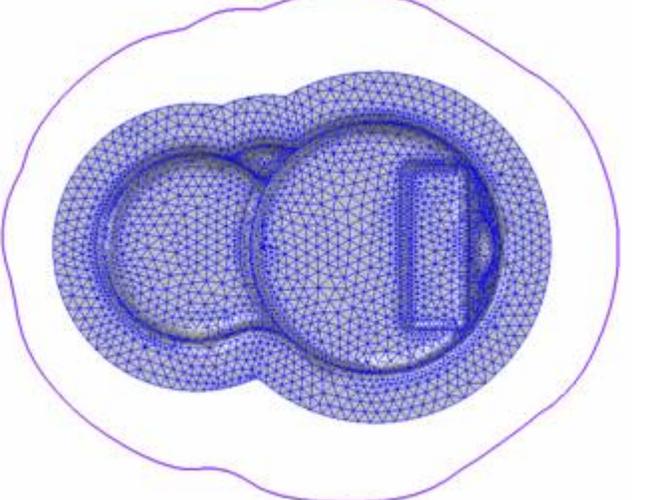
## 3DQuickForm Overview



3DQuickForm shares the same windows of SolidWorks and all the features can be accessed from 3DQuickForm Manager tree which is similar to the Feature Manager tree. You can use the menu system, or the 3DQuickForm Manager tree to manage analysis project. 3DQuickForm creates a folder in the 3DQuickForm Manager tree for each project. Sub folders define the parameters of the project.

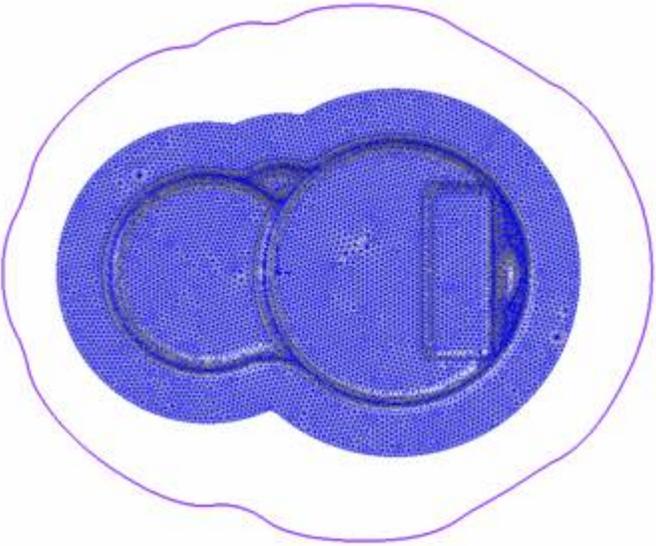
- Start a new project
- Part Definition
- Apply Boundary conditions
- Mesh the part
- Run the Project
- View the result
- Professional Function: Initial Blank
- Professional Function: Flanging

## Mesh size and accuracy

<p><b>Meshing Parameters</b></p> <p>✓ ✗ ?</p> <p><b>Basic Parameters</b></p> <p>Global Meshing Size 0.05</p> <p>Min. Meshing Size 0.02</p>	
<p><b>Meshing Parameters</b></p> <p>✓ ✗ ?</p> <p><b>Basic Parameters</b></p> <p>Global Meshing Size 0.025</p> <p>Min. Meshing Size 0.01</p>	

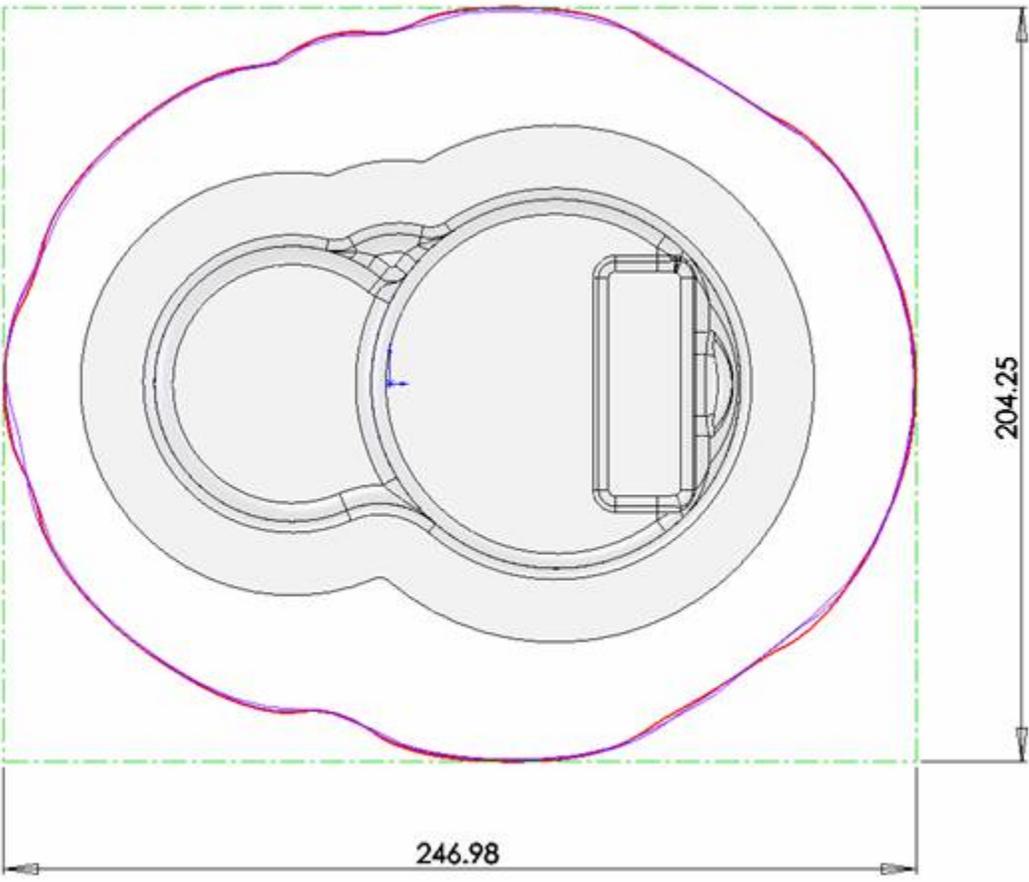
took 20s second to solve

took 45 seconds to solve

<b>Meshing Parameters</b>	
<p>✓ ✗ ?</p> <p>Basic Parameters ▲</p> <p>Global Meshing Size</p> <p>0.01</p> <p>Min. Meshing Size</p> <p>0.01</p>	

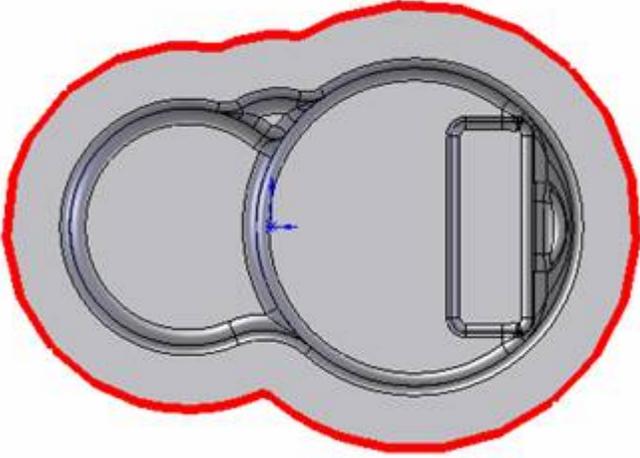
took 3 Minutes to solve

Red profile is the output by the finest mesh, the maximum deviation between the finest and the coarsest mesh is 2.3 mm; ~ 1% of the blank size



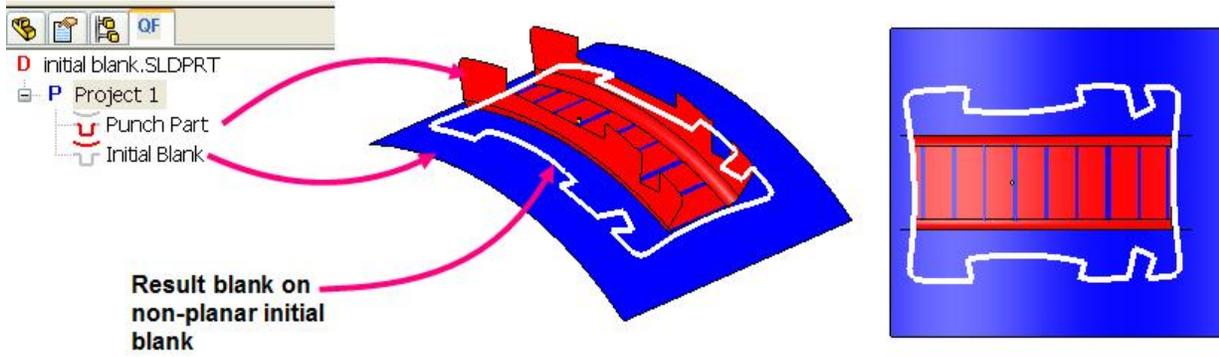
## Simulation fails

Simulation will fail under the following conditions:

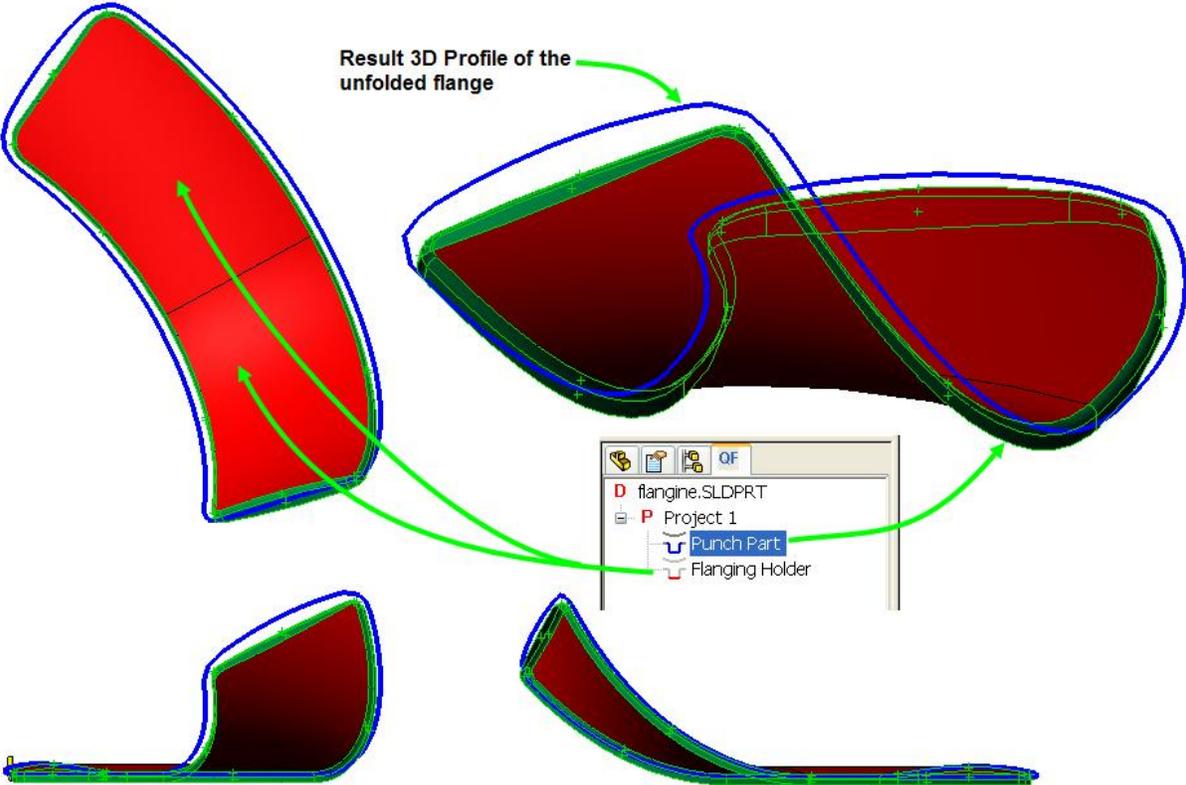
Failure reason	Syndrome
<p>Poor mesh quality</p> <ul style="list-style-type: none"><li>❖ A projected sketch will be created</li></ul>	
<p>Part complexity</p>	<p>Warning message for self-intersecting profile</p> 

# Initial Blank

Part can be unfolded to a non-planar initial blank



# Flanging



[www.3dquicktools.com](http://www.3dquicktools.com)

**QUICK  
FORM**



Unit 1711, Grand City Plaza, No. 1-17 Sai Lau Kok Road, Tsuen Wan, N.T., Hong Kong

Tel: +852 2788 2832 | Fax: +852 2777 6676 | Web: [www.3dquicktools.com](http://www.3dquicktools.com)

All Rights Reserved © 2012 3D QuickTools Limited