Validating wear simulations in heat exchanger plate stamping process through comparative analysis for enhanced productivity and quality

Donghui Fang[1], Dr.-Ing Kang Shen[2], Xiaolong He[2]

[1]Danfoss (Jiaxing) Co., Ltd

[2]Ansys Germany GmbH

1 Abstract

Tool wear is a common problem in the manufacturing process of heat exchanger plates. The shape of HP changes due to tool wear, causing issues in subsequent processes such as welding, brazing, and assembling. In this paper, a simulation model of heat exchanger plates using Ansys Forming® is created to predict tool wear. It provides guidance for the shape design of heat exchanger plates, aiding in the planning of tool repair and replacement.

2 Introduction

As a key tool in the production process of heat exchanger plates, the condition of the forming tool directly affects the quality and performance of the heat exchanger. Especially after large-scale stamping, wear can cause changes in the geometric dimensions and shape of the plate, resulting in a decrease in overall performance, shortened service life, and leakage problems of the heat exchanger.

There are multiple factors having effect on wear, including contact pressure, relative sliding speed, environmental parameters as humidity and temperature [1], mechanical properties of tool metal as tensile strength, hardness, protective coating, surface roughness [2].

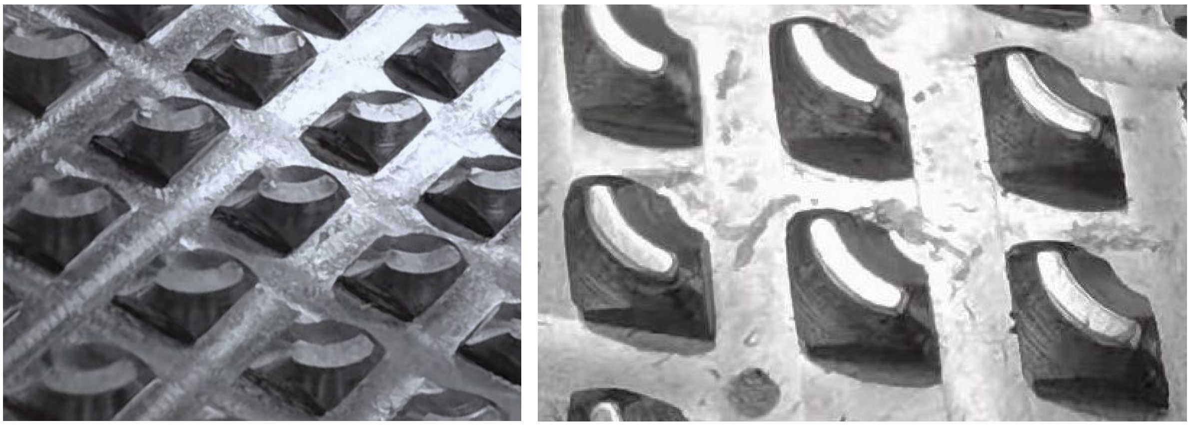
In this paper, the study employs Ansys Forming to simulate the influence of wear in the sheet metal forming process. The simulation model incorporates wear prediction and automatically updates the geometry based on the wear depth. The theoretical basis is taken from [3].

Finally, the wear analysis module was applied to the wear calculation of a heat exchanger plate. This enables manufacturers to proactively maintain and replace tools and allows tool designers to improve and optimize tool design.

3 Industry practice

From past experiences, tooling wear can cause many inconveniences such as unqualified product size, affecting brazing quality, false brazing of heat exchangers, seriously affecting overall product performance and reliability, reducing production efficiency, and increasing maintenance costs. It is often necessary to wait for tool wear to occur, passively detect tool size to judge tool status, and determine whether replacement and maintenance are needed. This process often has a significant lag. Various dimensional requirements are proposed during the product design phase, such as height tolerances for dimples and top area tolerances for brazing joints. Due to the particularity of the micro channel heat plate, real-time monitoring of tool feature dimensions is also difficult. Therefore, understanding the mechanism of tooling wear and developing a tool that can predict tooling wear, and the life cycle of the tooling is urgently needed.

Figure 1 shows the tool geometry in manufacturing. After 100,000 cycles of stamping, the edge of the dimple becomes worn. As a result, the stamped part does not have a sufficiently flat top surface for brazing. In practice, this tool needs to be replaced with a new one to ensure product quality.



a) tool after 2000 cycles of stamping

b) tool after 100,000 cycles of stamping

Figure 1 Bottom tool for heat exchanger plate

4 Model preparation

The first step is to set up the model in Ansys Forming. In this paper, a small-scale model is adopted. It contains two tools – an upper tool and a lower tool. The lower tool is fixed during the drawing process, while the upper tool moves to the home position with displacement control.

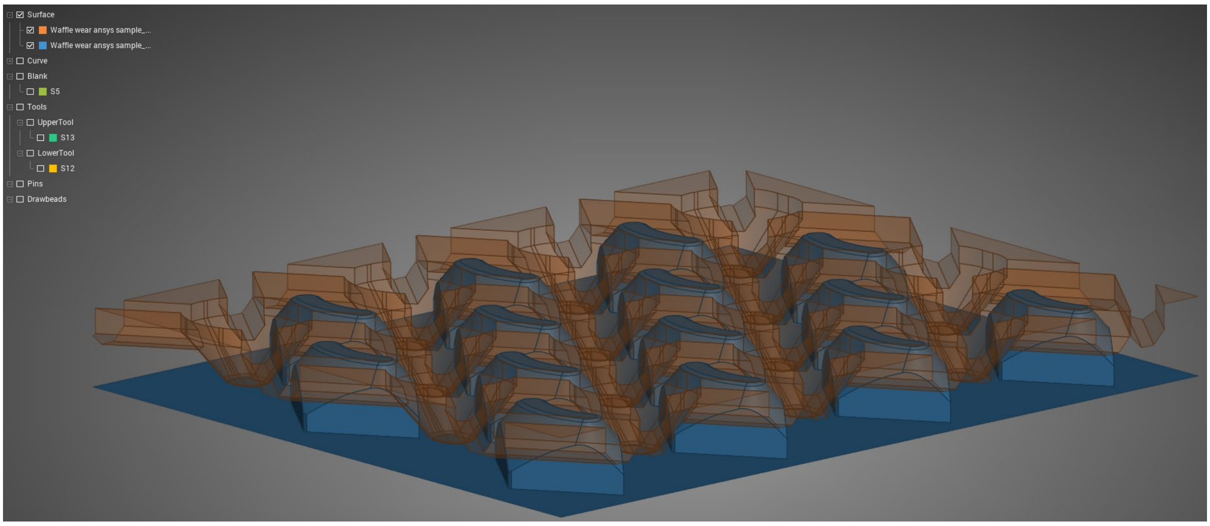


Figure 2 crash form simulation in Ansys Forming

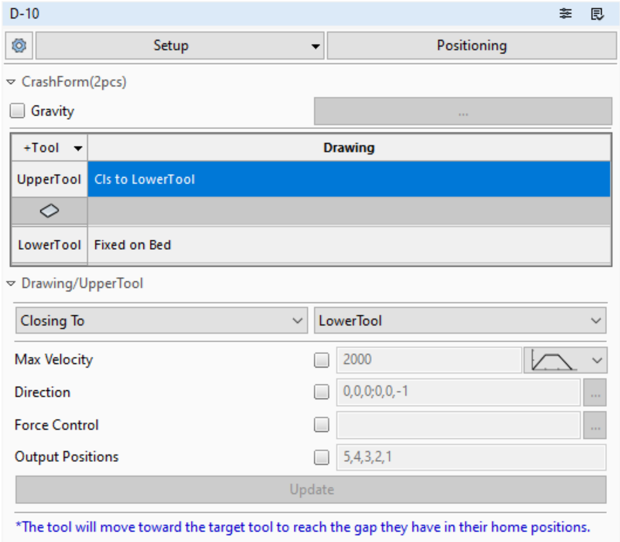


Figure 3 define tool operation

In simulation, the blank material is stainless steel 316L and the tool material is ASSAB 88. The tool deformation during the drawing process is neglected and tool is assumed as rigid body with a contact wear.

For the contact wear, Archard's wear law is applied. The wear rate at a contact point is defined as

$$\dot{\omega} = k \cdot \frac{p\dot{d}}{H}$$

$\dot{\omega}$ is the wear depth evolves with time. k is a scale factor. p is the contact interface pressure. $\dot{d}$ is the relative sliding velocity of contact points in contact. H is the hardness of the surface [4].

Ansys Forming GUI in 24R1 does not support contact wear definition. Keywords about contact wear need to be added after entering the Runner. The needed keywords are *CONTACT_ADD_WEAR, which defines wear model to a contact interface.

```
*CONTACT_ADD_WEAR
$#     cid     wtype        p1        p2        p3        p4        p5        p6
         1         0    5.0e-5       650      2020       0.0       0.0       0.0
```

Then, contact between tool and blank need output the wear depth. Contact variable SBPR is set to 2, so that the wear result with be included in dynain file with keyword *INITIAL_CONTACT_WEAR.

```
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE
$#     cid                                                                 title
         1
$     SSID      MSID     SSTYP     MSTYP    SBOXID    MBOXID      SAPR      SBPR
         1         3         2         2         0         0         0         2
$       FS        FD        DC        VC       VDC    PENCHK        BT        DT
      0.12       0.0       0.0       0.0      20.0         0     0.01.00000E20
$      SFS       SFM       SST       MST      SFST      SFMT       FSF       VSF
       1.0       1.0       0.0    -0.001       1.0       1.0       1.0       1.0
```

With the setting of contact, the output of post-processing need to be set too. In one drawing simulation, the information of wear depth will be calculated. This wear depth could be very small, and no significant geometrical change could be observed. Then, an Assumption is that one drawing simulation holds the contact states for several drawing operations in real tests. Then the wear depth will be calculated for several drawing operations before we change the geometry. This assumption is valid only if the wear depth is much lower as manufacturing tolerance (0.05mm in this case). To apply this assumption, the variable NCYC in *INTERFACE_SPRINGBACK_LSDYNA is set.

```
*INTERFACE_SPRINGBACK_LSDYNA
$#    psid      nshv     ftype         _    ftensr    nthhsv         _   intstrn
         1       100         0         0         0         0         0         0
$#    optc      sldo      NCYC    fsplit
OPTCARD            0      1000         0
```

In this simple case, no further operation like trimming and springback is needed for subsequential simulation, because only tool wear is focused. Only wear information should be saved in dynain file for next drawing simulation. The rest of the output could be excluded with

```
*INTERFACE_SPRINGBACK_EXCLUDE
BOUNDARY_SPC_NODE
NODE
ELEMENT_SHELL
INITIAL_STRESS_SHELL
INITIAL_STRAIN_SHELL
```

Optionally, the NWEAR variable in *DATABASE_EXTENT_INTFOR could be set. Then the wear depth and sliding distance could be output in INTFOR file.

```
*DATABASE_EXTENT_INTFOR
$#   nglbv     nvelo    npresu    nshear    nforce     ngapc     nfail    ieverf
         1         1         3         1         1         1         0         0
$#   NWEAR
         1
```

After solving one drawing simulation, the wear information could be found inside of dynain file:

```
*INITIAL_CONTACT_WEAR
$#     cid       nid    wdepth        nx        ny        nz      iseq      ncyc
         1    842841 4.818E-10-2.909E-03-1.408E-01-9.900E-01         1      1000
       ...
```

In this result, wear depth depth is provided with a direction nx, ny and nz. This initial wear information could be applied on tool geometry. Here we have two questions:

1. How to decide the value of NCYC? The assumption is valid only if the wear is small in drawing simulation. Before running a wear simulation, the wear depth is unknown.
2. Wear depth is defined for contact nodes. With a fine feature in heat exchange plates, there are multiple surfaces with tiny radii. The contact nodes on that radii could be worn faster than other nodes and have a significant geometrical change. Then, it could lead to poor mesh.

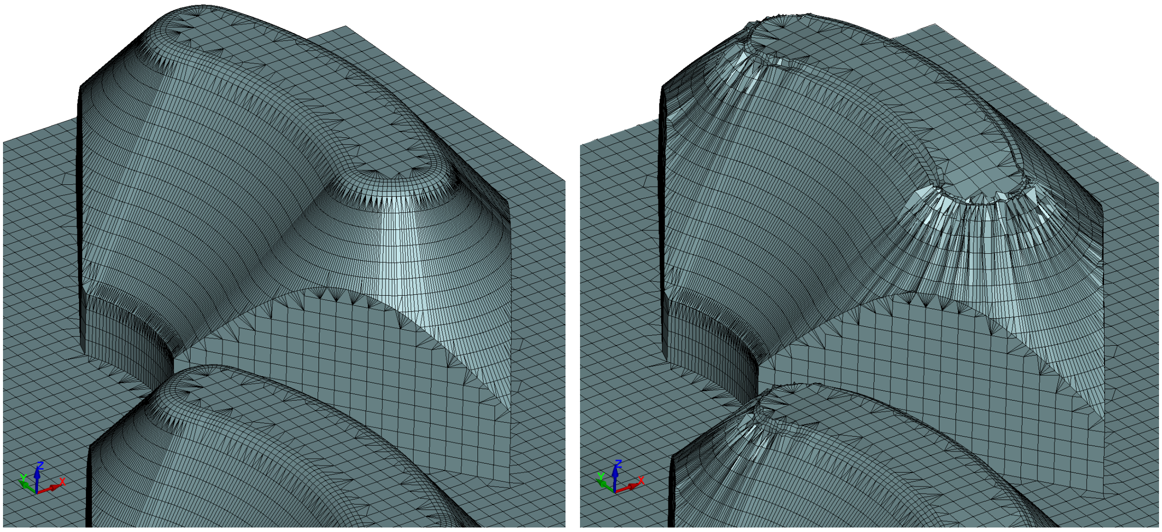


Figure 4 traditional tool mesh before and after wear simulation

In [3], it is proposed that using wear application of LS-PrePost. In there, a maximum wear distance and a smooth function is provided. In this paper, same method is applied with a LS-PrePost script.

```
openc keyword " sim.dyn"
import keyword "D-10.dynain"
wear maxdist 0.010000
wear compute
```

```
wear smooth
wear accept
output " tool_mesh.k" 1 1 1 1 1 1 0 0 0 0 0 0 0 0 0 1.000000 0 0
exit
```

Furthermore, the latest feature “Adaptive mesh regeneration” is applied in Ansys Forming. This feature is currently located in the Mesh Check panel. It requires a defect-free input mesh, so it’s recommended to apply mesh check before refining. It produces a mesh with only quadrilateral elements in interior. It performs best on geometries with smooth gradients.

With this feature, small triangle elements are replaced with smooth quad elements shows in following picture.

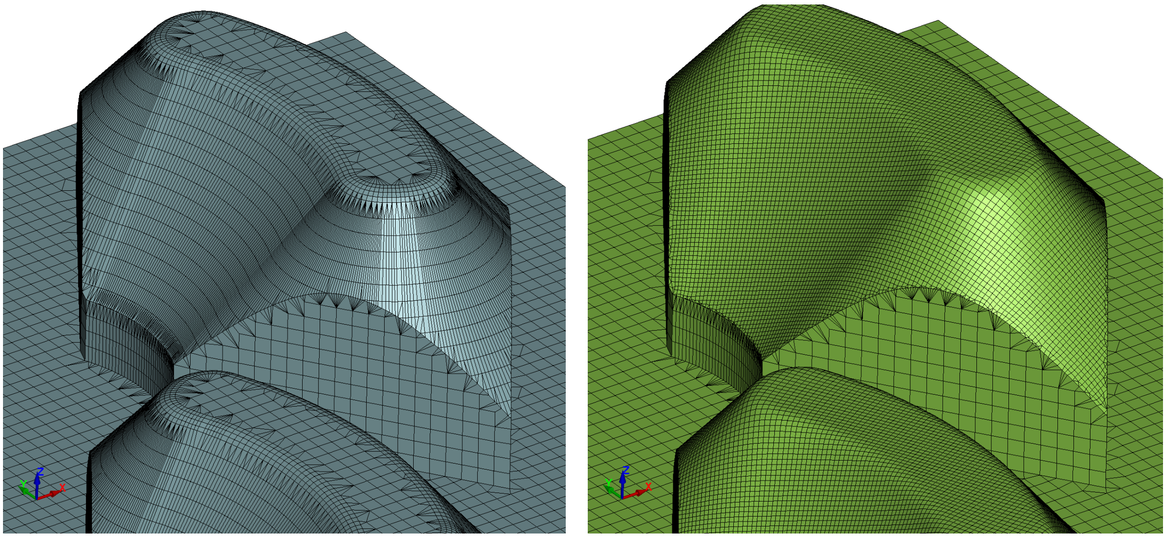


Figure 5 enhanced mesh quality after mesh regeneration

The process of stamping, wear, smooth the surface and accept new geometry needs to be done multiple time to mimic the wear in practise. A batch file is created to support whole process.
call "lsdynamsvar.bat"

```
for /l %%x in (1, 1, 100) do (
   echo %%x
   copy "tool_mesh.k" "tool_mesh_%%x.k"
   mpiexec -c 10 -aa -a "ls-dyna_mpp_s_msmpi.exe" i=sim.dyn memory=120m
   " lsprepost4.11.exe" c=smooth.cfile -nographics
   echo %%x
   timeout 2
)
```

In this example of batch, 100 times stamping process is performed. All tool mesh files during the process are kept for result check.

One stamping process takes about 90 seconds. The whole wear simulation with 100 iteration takes in total 2.5 hours.

5 Results

In this simulation, the maximum wear depth is controlled by 0.01 mm in the script of LS-Prepost. With NCYC=1000 there is no wear depth larger than 0.01 mm. It means with 100 iterations; the wear simulation has accumulated wear for 100k cycles. Figure 6 shows the initial geometry, worn geometry of 10k, 20k and 100k cycles.

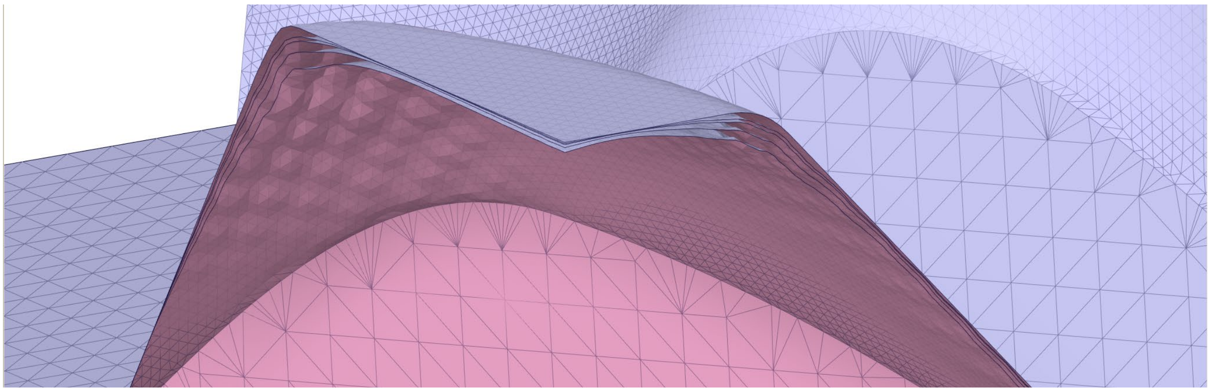


Figure 6 initial tool [top], tool after 10k, 20k and 100k cycles [bot]

The scanned tool geometry of tool sample is imported and shows a good fit with simulation. The maximum wear depth is 0.025 mm and the difference between scanned tool and simulation is 0.005 mm

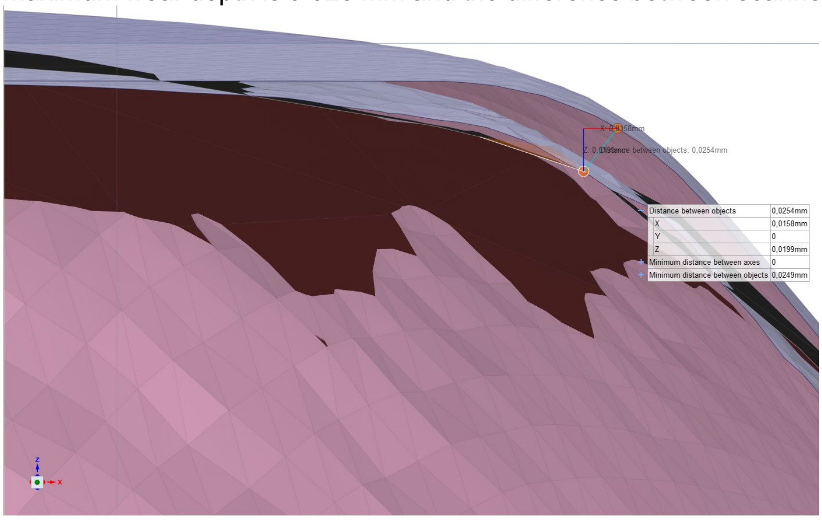


Figure 7 wear depth from inital toll

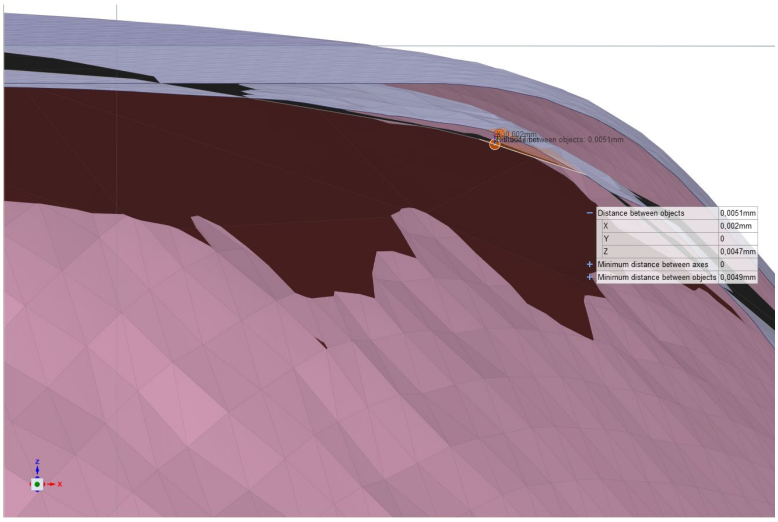


Figure 8 compare wear surface with scanned part [black] and simulation result.

The direct impact of wear is changing the area of flat zone for brazing. Following figure shows the change of the surface. By calculating the surface (after smoothing of worn face), it lost 44% of its flat area.

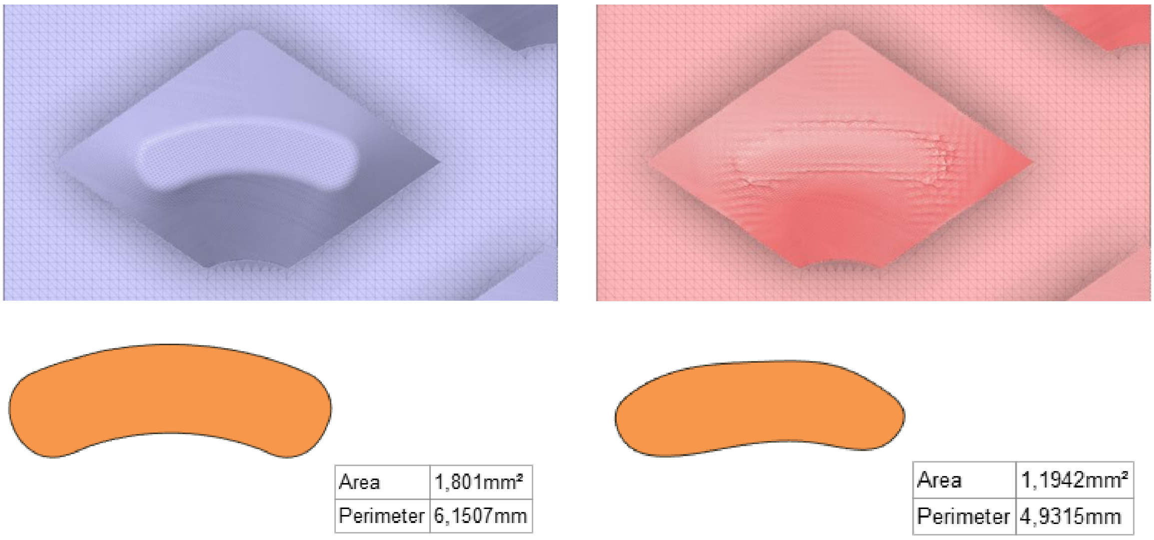


Figure 9 area chage in Brazing zone

Tool wear led to a lower quality of manufacturing. In this case, the shape of heat exchange plate changes significantly after 100k wear cycles. The flat surface length decreases from 2.02mm to 1.26mm (38%).

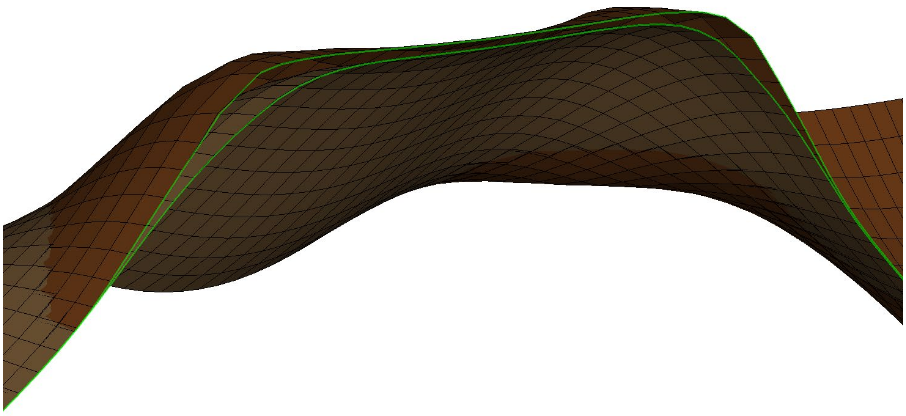


Figure 10 Heat Exchange Plate before and after tool wear

6 Summary

Manufacturers of sheet metal products require simulations to predict tool wear. Ansys Forming offers a comprehensive workflow that integrates advanced wear simulation into stamping processes, delivering accurate results. Using Ansys Forming, tool designers for heat exchanger plates can utilize simulation outcomes to guide tool design and enhance robustness against wear. This workflow enables manufacturers to predict the lifespan of stamping tools and prepare for potential repairs or replacements.

7 Literature

[1] Dong, H. Tribological properties of titanium-based alloy. In Surface Engineering of Light Alloys; Dong, H., Ed.; Woodhead Publishing: Sawston, UK, 2010; pp. 58–80.

[2] Trzepieciński, T. Tarcie i Smarowanie w Procesach Kształtowania Blach; Wydawnictwo Naukowe PWN: Warszawa, Poland, 2023.

[3] Borrvall, T., Jernberg, A., Schill, M, Deng, L., Oldenburg, M., Simulation of Wear Processes in LS-DYNA – 14th international LS-DYNA Users Conference

[4] LS-DYNA Keyword User's Manual Volume I, R15