Powering Innovation That Drives Human Advancement

Ansys Forming – Accurate, Scalable, Process-Driven GUI for Metal Forming Simulation

Xinhai Zhu

Agenda

- Introduction of Ansys Forming
  - History review
  - Ansys Forming GUI
- Typical applications
  - Multi-step forming
  - Clamping
  - TWB simulation
  - Springback compensation
  - Trimming curve development
  - One step method
- Key functions
  - Solid element in stamping simulation
  - Mesh check/repair
  - Mesh regeneration
  - Formability Index
  - 3D drawbead generation of bead force prediction
  - Non-linear contact
  - Variable friction
  - New universal material model
  - Reconfigurable settings
- Exciting future

Introduction

Background

- LSDYNA has long been used in stamping simulation
  - LSDYNA was one of the first commercial FEA software for stamping simulations
  - Has built a strong reputation for its accuracy, especially in springback prediction
  - Used to rely on third-party GUI for stamping simulation.
- Ansys Forming is a dedicated package to stamping simulation
  - Integrated GUI - Pre, solver and post processing
  - A Designer’s tool
  - Can simulate the entire stamping process within a single platform
  - Streamlined Formability/Trimline development/Springback Compensation functions.

Design Objectives

Easy to use - Friendly user interface for intuitive use.

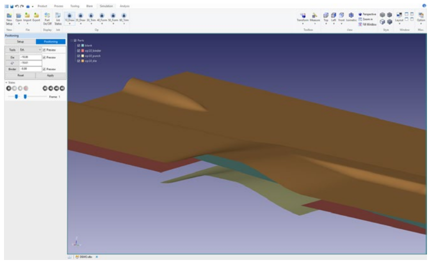
Efficient - Smart adaptivity, contact auto move, new material algorithm, scalability through MPP, in- core technology.

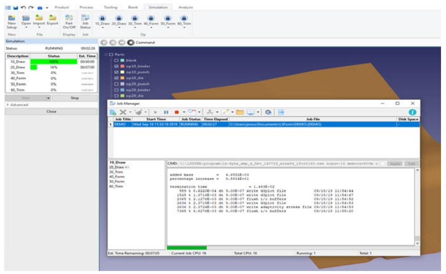
Robust - automatic normal check for contact, optimized setting, consistent solution.

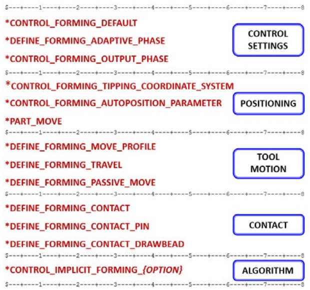
Fully Integrated Platform

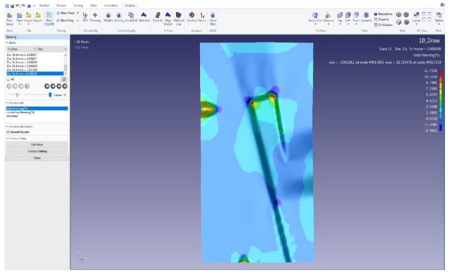
- Ansys Forming provides a unique platform which has a seamless fully integrated GUI with pre-post processing and uses LS-DYNA as a solver.
  - Easy to setup multi-stage forming simulations
  - Customizable template-based method allows user to easily define different forming processes
  - A job-submitter allows user to run the job easily
  - User can seamlessly evaluate simulation results when the job is running

Pre-Processing	Job Submission	LS-DYNA Solution	Post-Processing
Model Setup	Job Runner	New Forming Keyword Features	Analysis







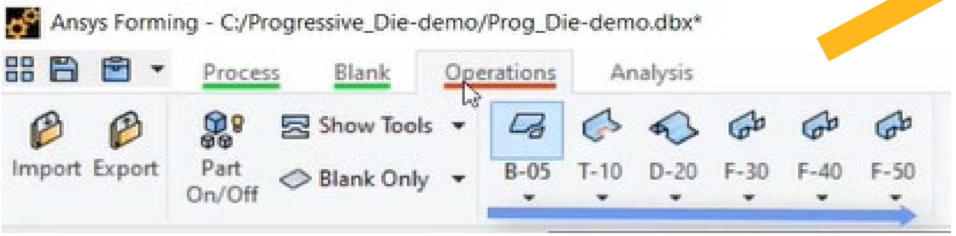


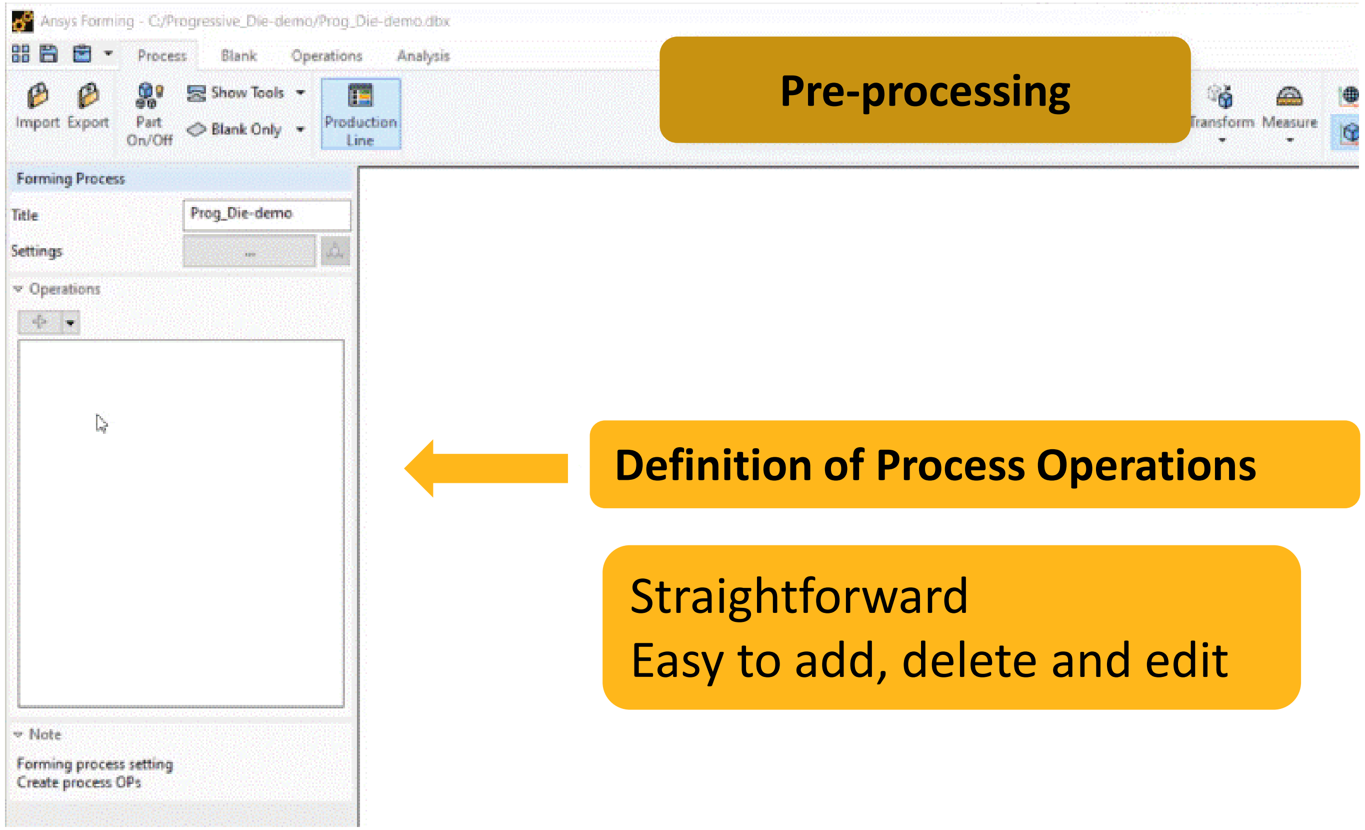
Process-based Workflow

During process design/verification, engineers must perform the simulations of drawing and secondary operations, such as trimming, flanging and restriking, as well as springback.

Process-based workflow is the key to analyze and optimize all the operations of a stamping process.

Straightforward Operation Definition
Easy to use: ADD, DELETE & EDIT





Pre-processing

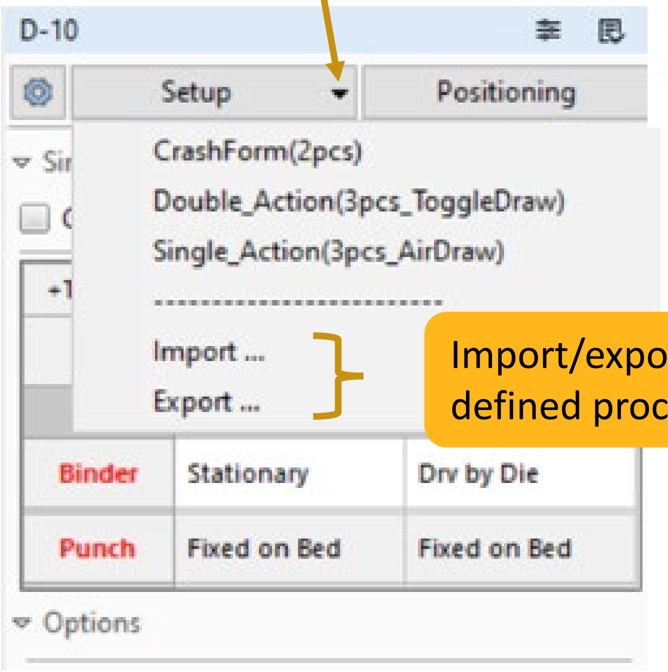
Definition of Process Operations

Straightforward
Easy to add, delete and edit

Preset/User-Defined Process Templates

Convenient to set up a typical process
Guidance for new forming users
Easy to edit for a complicated process

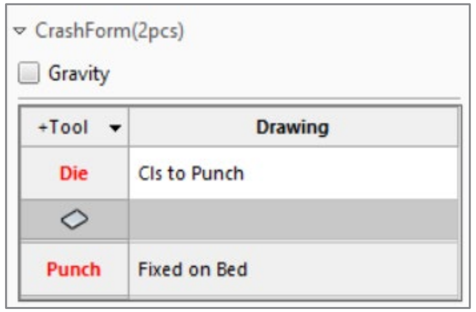
Drop down for a preset or user-defined process template

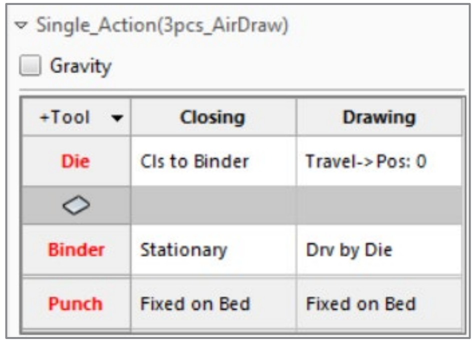


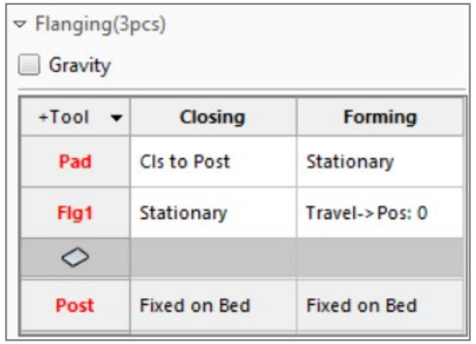
Import/export a user-defined process template

Preset Process Templates

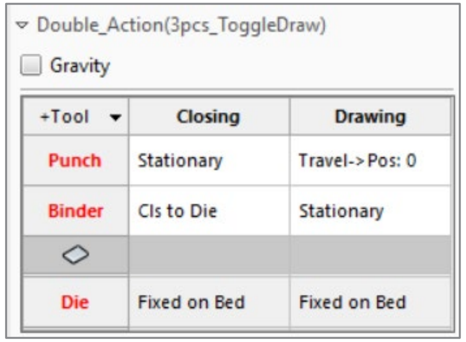
Draw operation



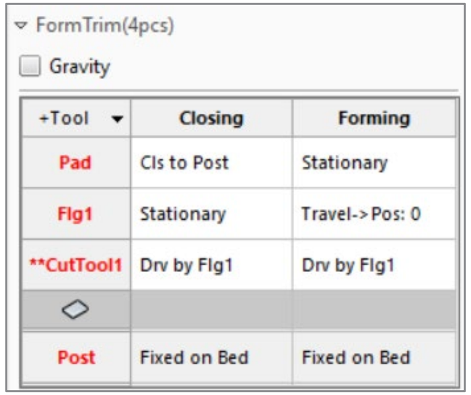




Flanging/Restrike



FormTrim



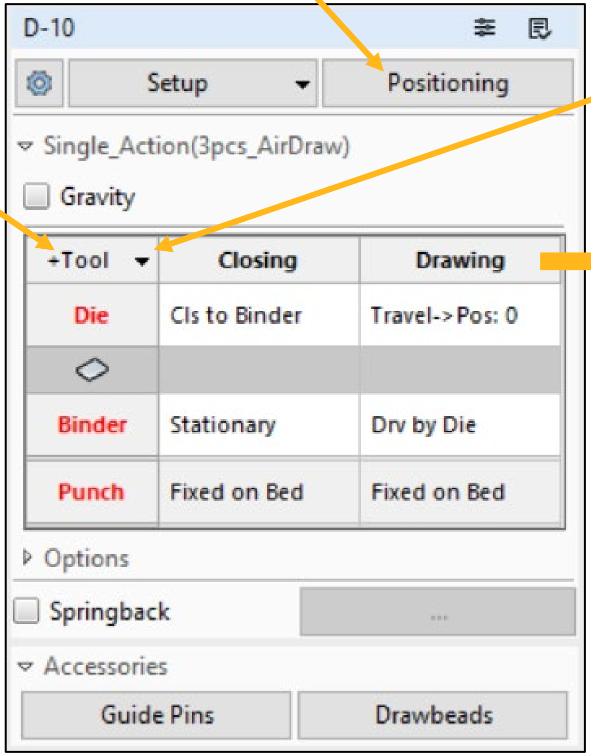
Innovative Tabular Tooling Setup

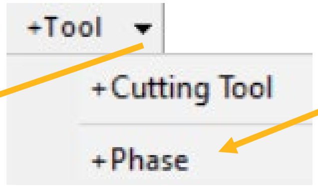
Flexible, easy and straightforward

- Flexible to define complicated tool motion
- No limitation on total tool number
- Easy to manage the tools and accessories
- Integrated with gravity and springback options

Preview/adjustment of tool positioning

Click ‘+ Tool’ to add a new tool





Drop down from ‘+ Tool’ and click ‘+ Phase’ to add a new motion phase

The first column is the tool list

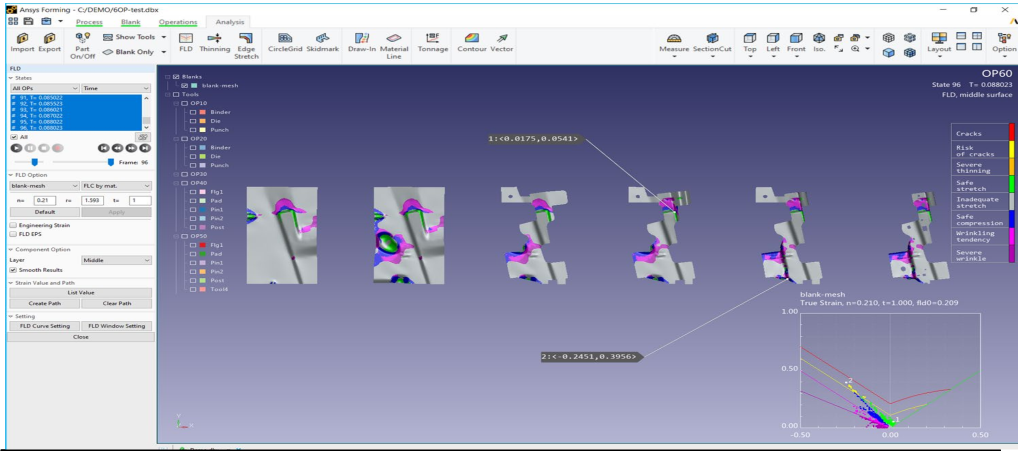
The first row is the motion phase list specifying the tool motion phases in current operation stage

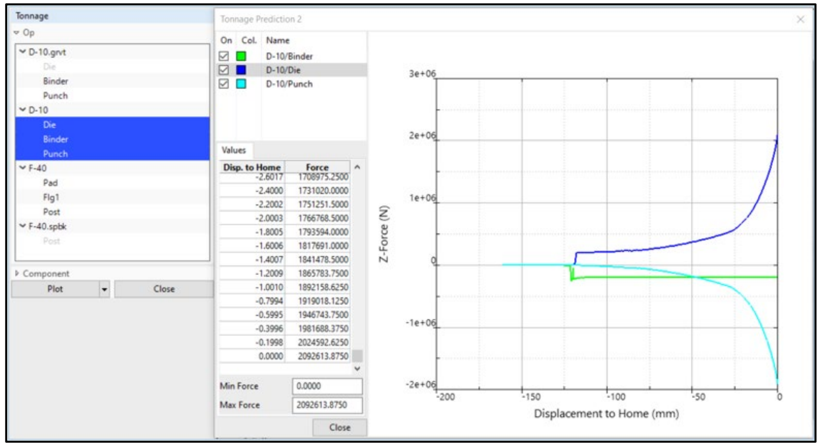
(e.g., a binder-closing phase and a drawing phase for a 3-piece air draw process)

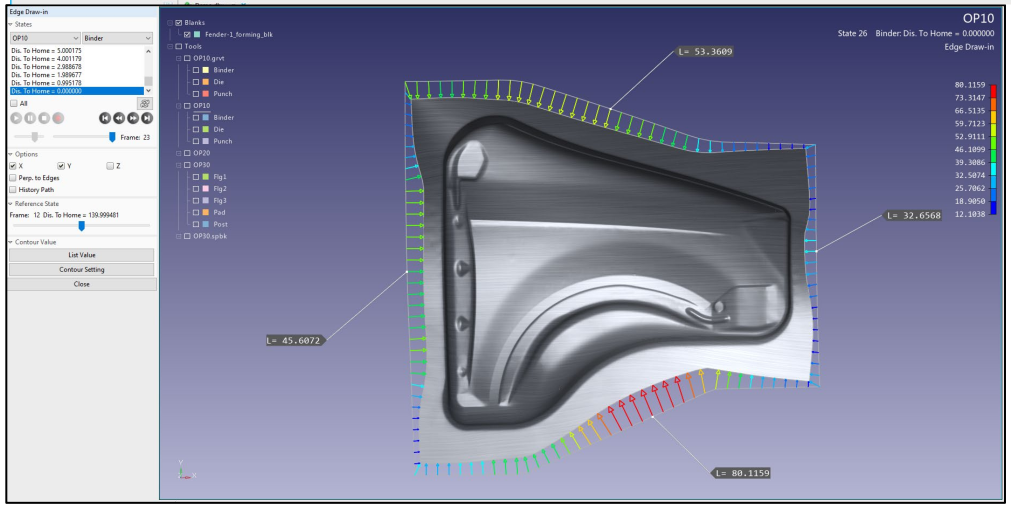
Ansys

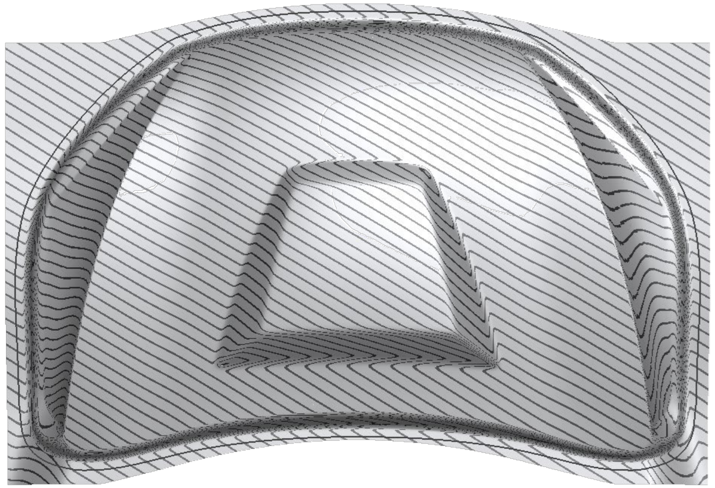
Provide all the necessary post processing functions

- ❑ Intuitive GUI
- ❑ Special forming modules
- ❑ Integrated post-processing of multi-stage jobs
- ❑ Easy multiple-window management
- ❑ Up-to-date graphic rendering
- ❑ High software stability





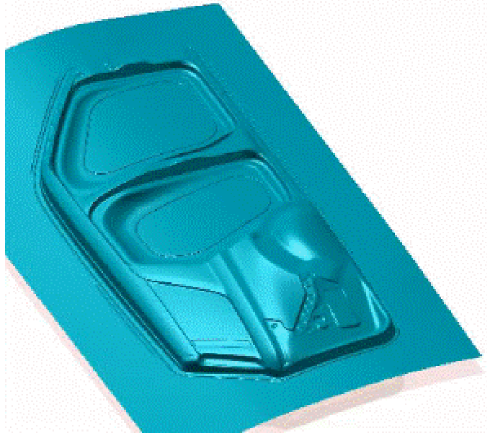




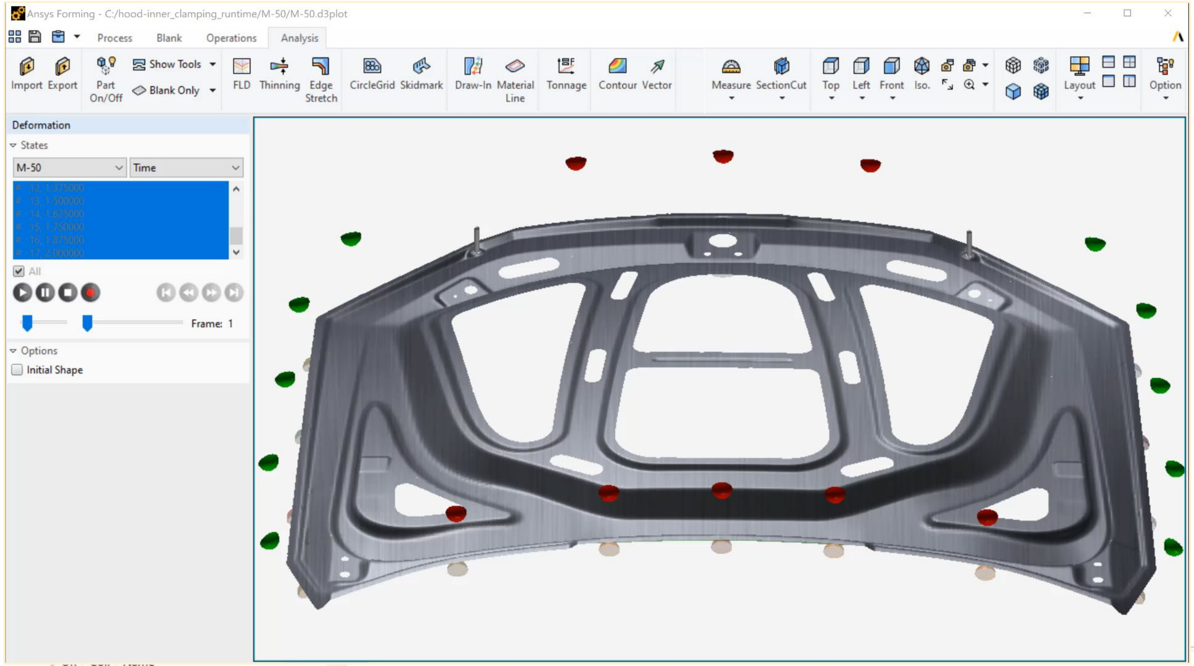
Typical Applications

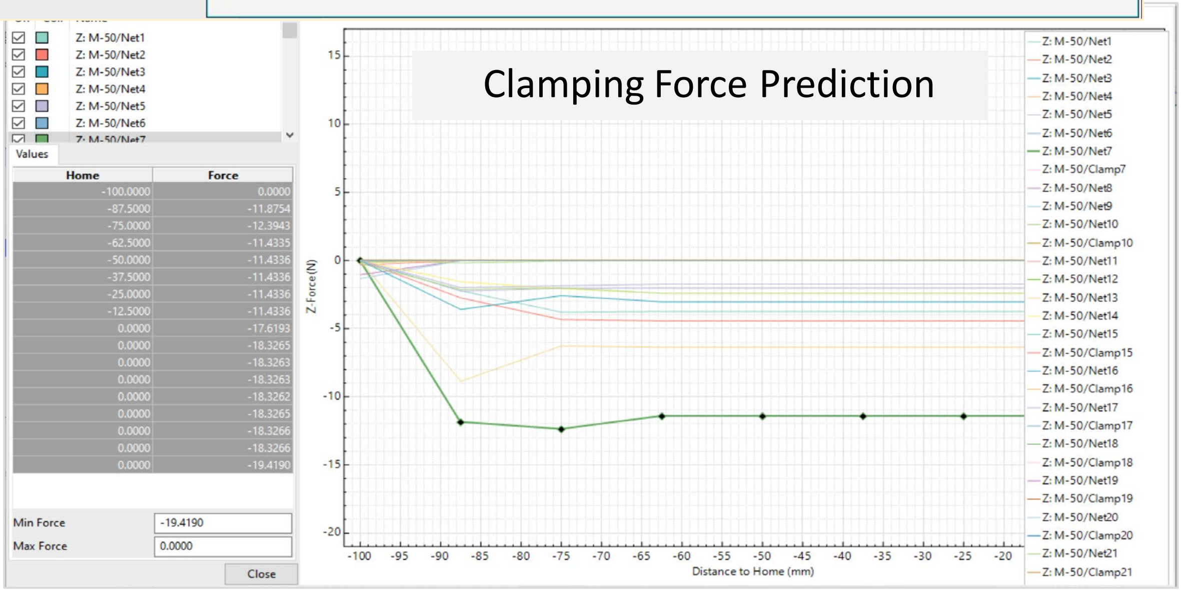
Traditional stamping simulation

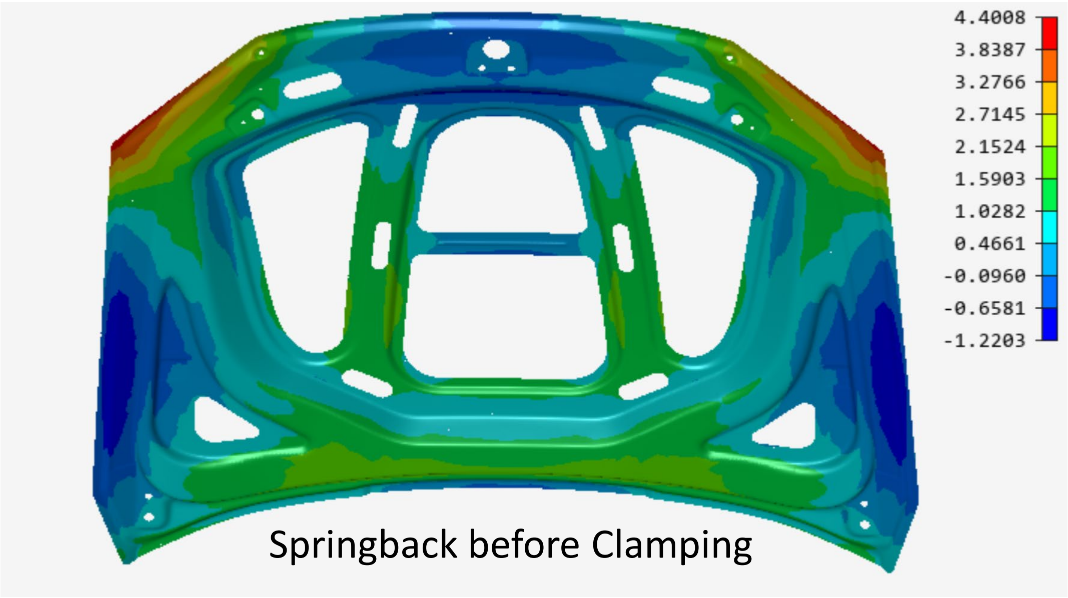
- Simulation of entire stamping process
  - blanking
  - Gravity loading
  - Draw
  - Trimming
  - Flanging/restriking
  - Spring back

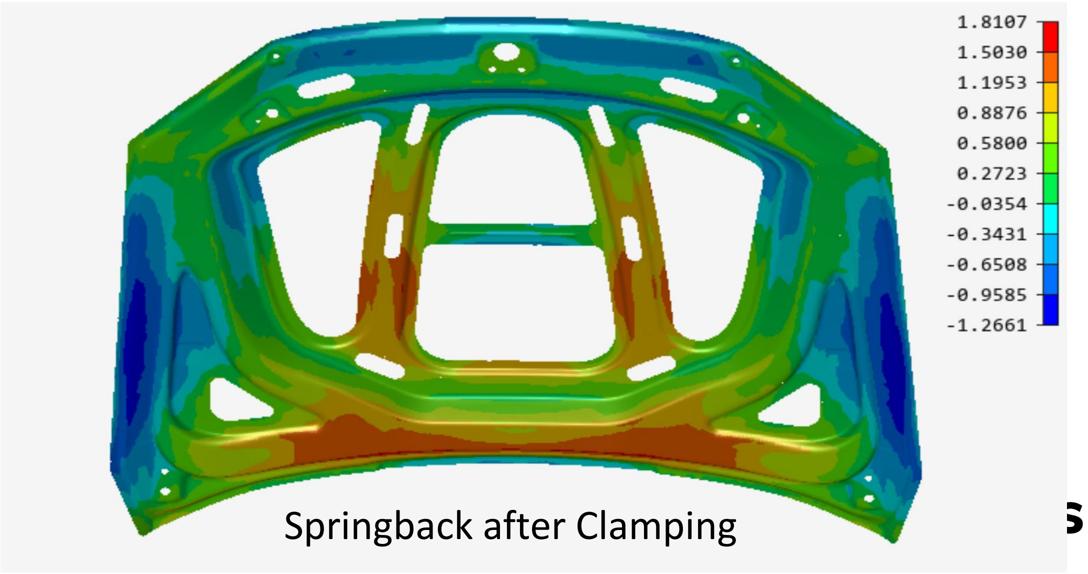


Clamping Simulation



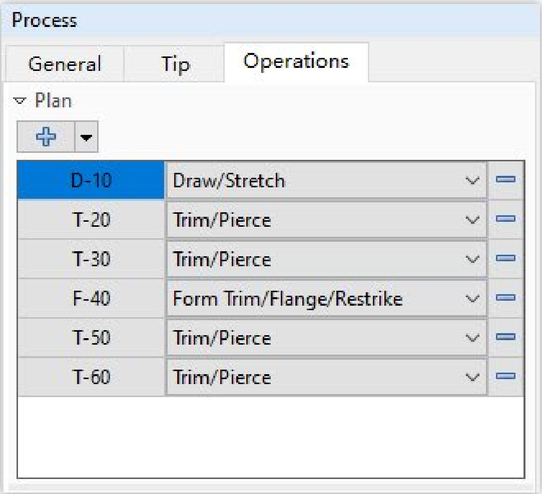


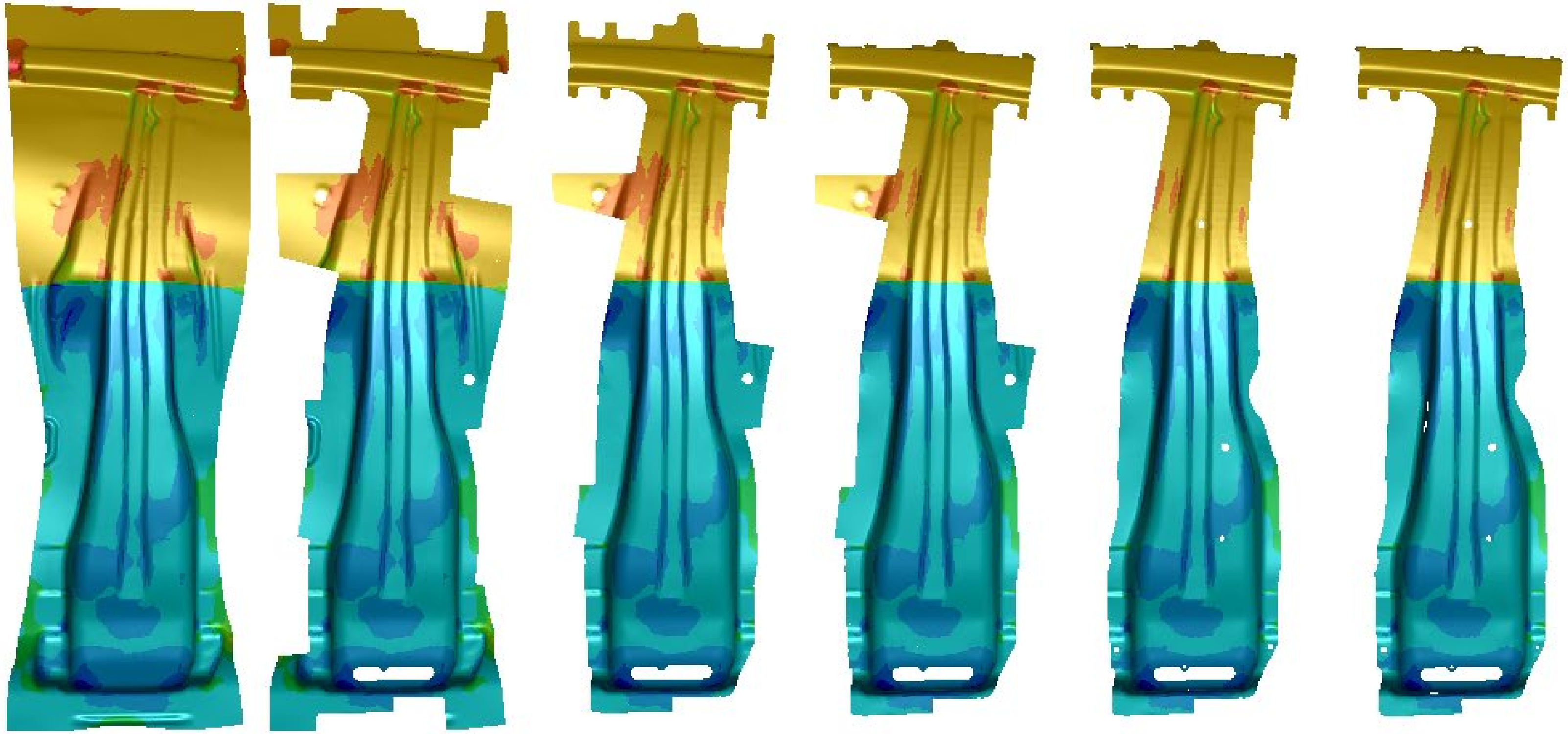




Tailor Welded blank

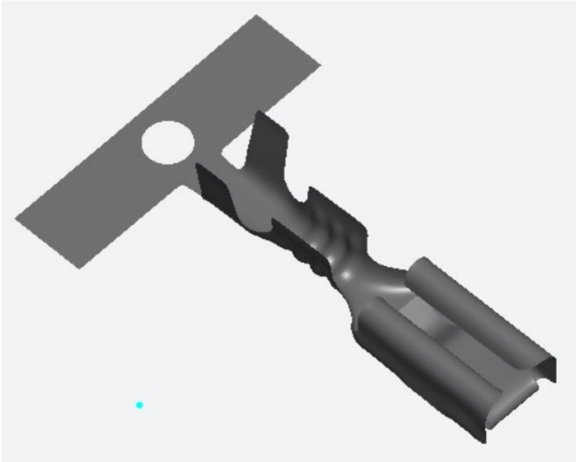
- 2025R2 enables the creation of tailored blanks with multiple weld lines

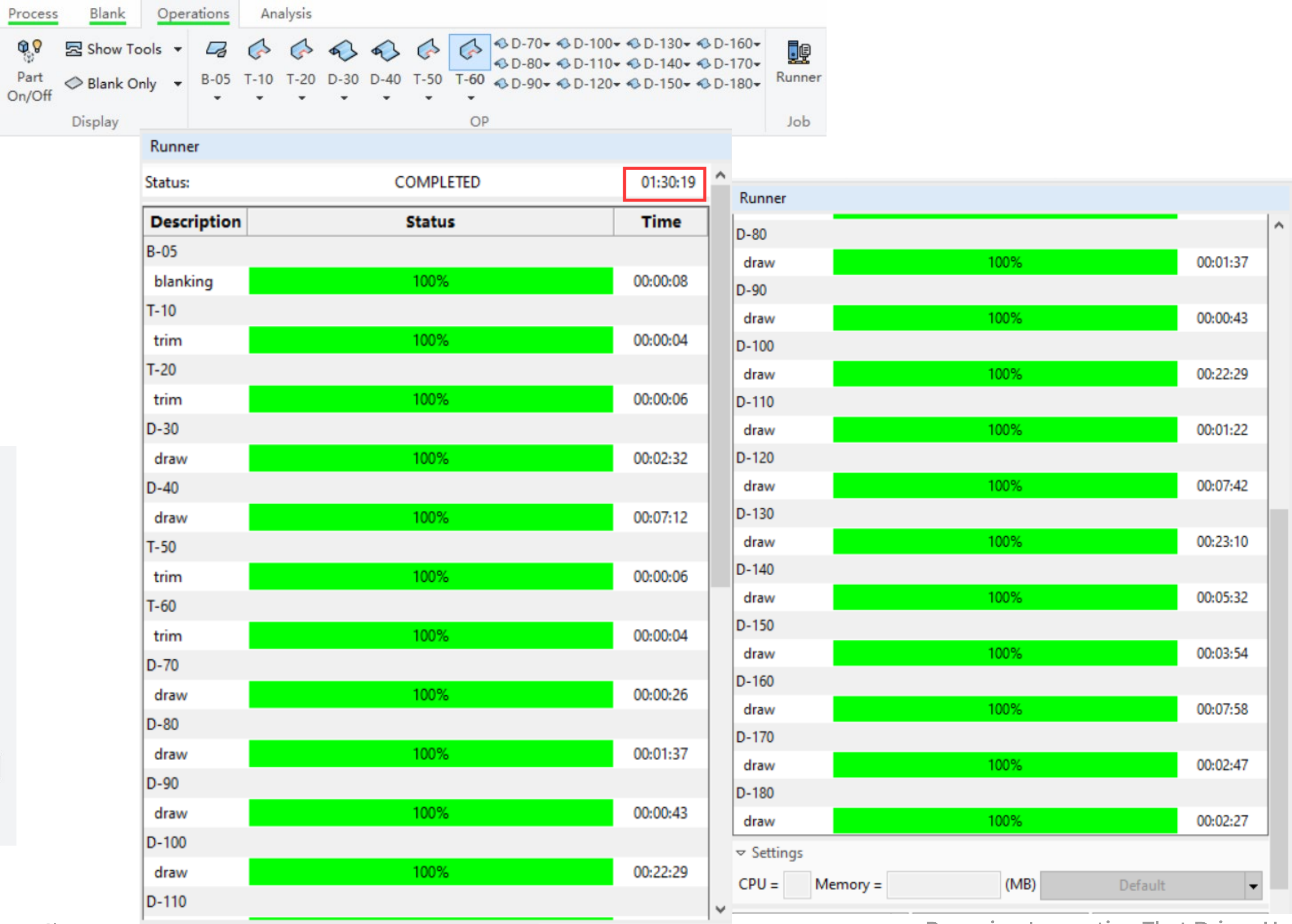




Progressive Die process

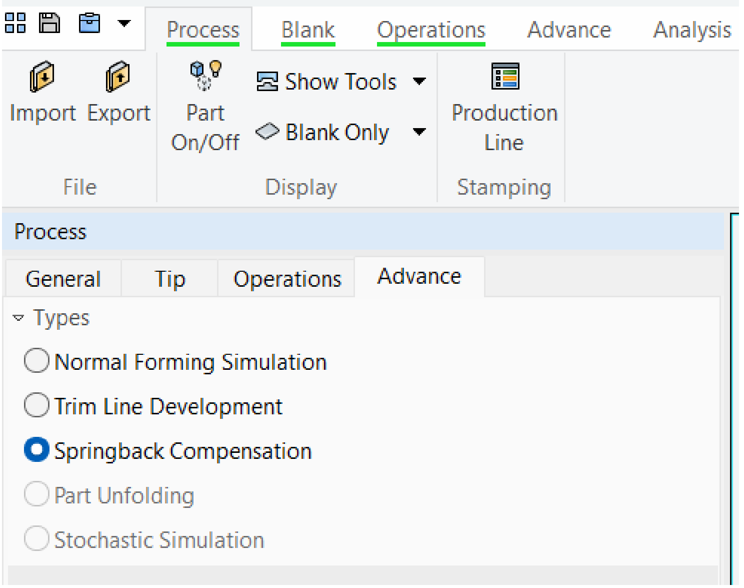
- The process involves 20 stations

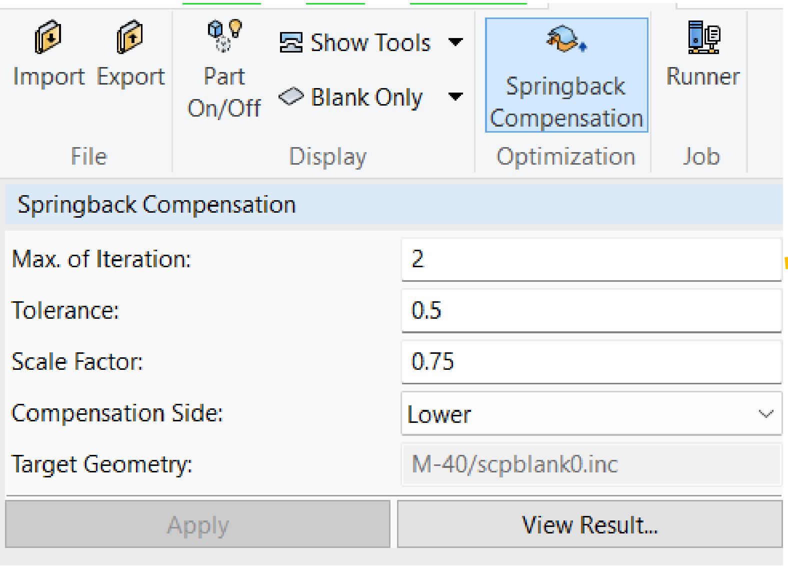




Springback Compensation

- The springback compensation provides a workflow to correct the springback deviation
  - Iterative method
  - Fully automatic
- Springback and draw forming can have different coordinate system





Maximum number of iterations

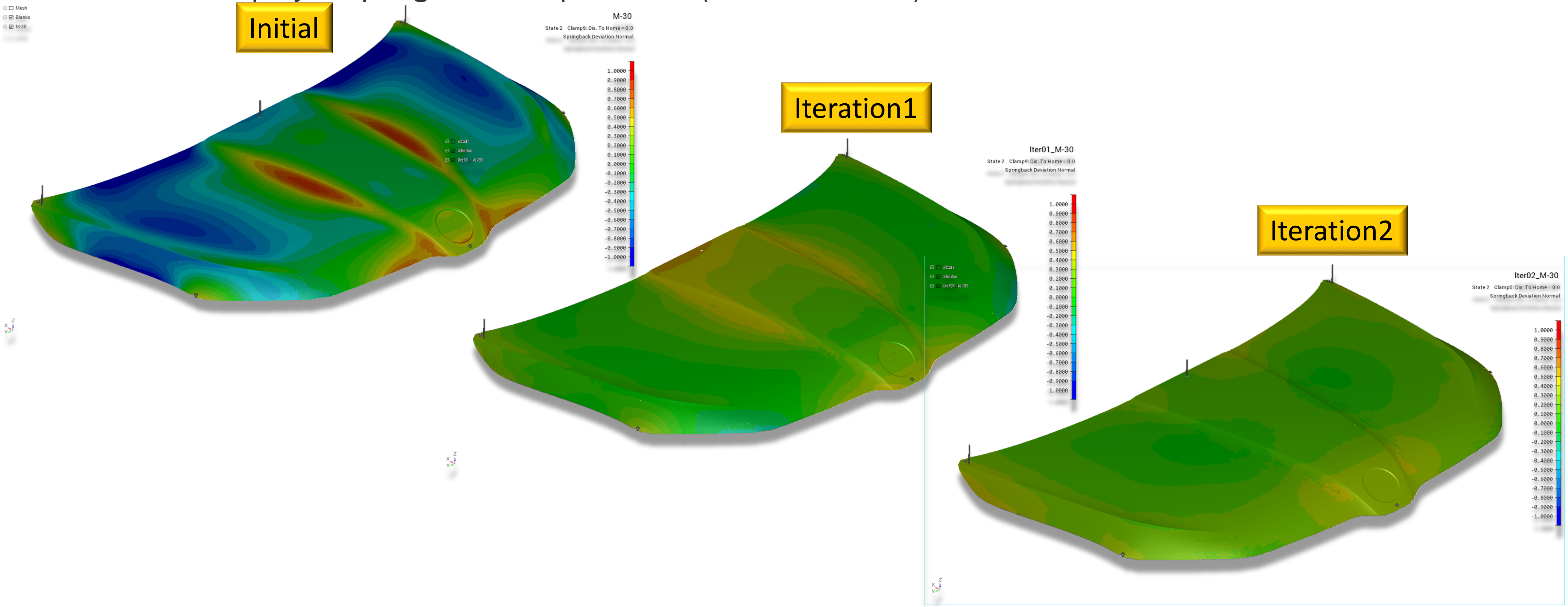
Maximum difference between target and final part geometry

Scaling of the compensation (0-1). Scaling equals 1 will compensate faster

Point to the side of the tools to be compensated

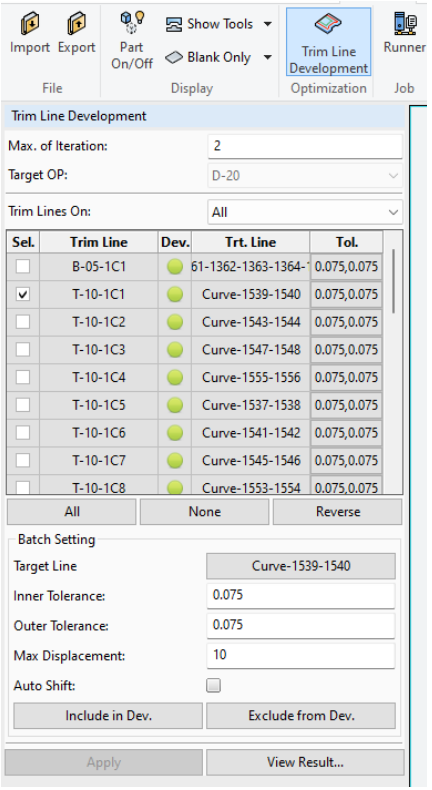
Springback Compensation

- **Postprocessing**
  - Result display of springback compensation (Blank deviation)



Trimming Curve Development

- The user interface facilitates setting up the development of trimming curves
- For every iteration, the user can evaluate the resulting part geometry to the target boundary



Run trimming curve development

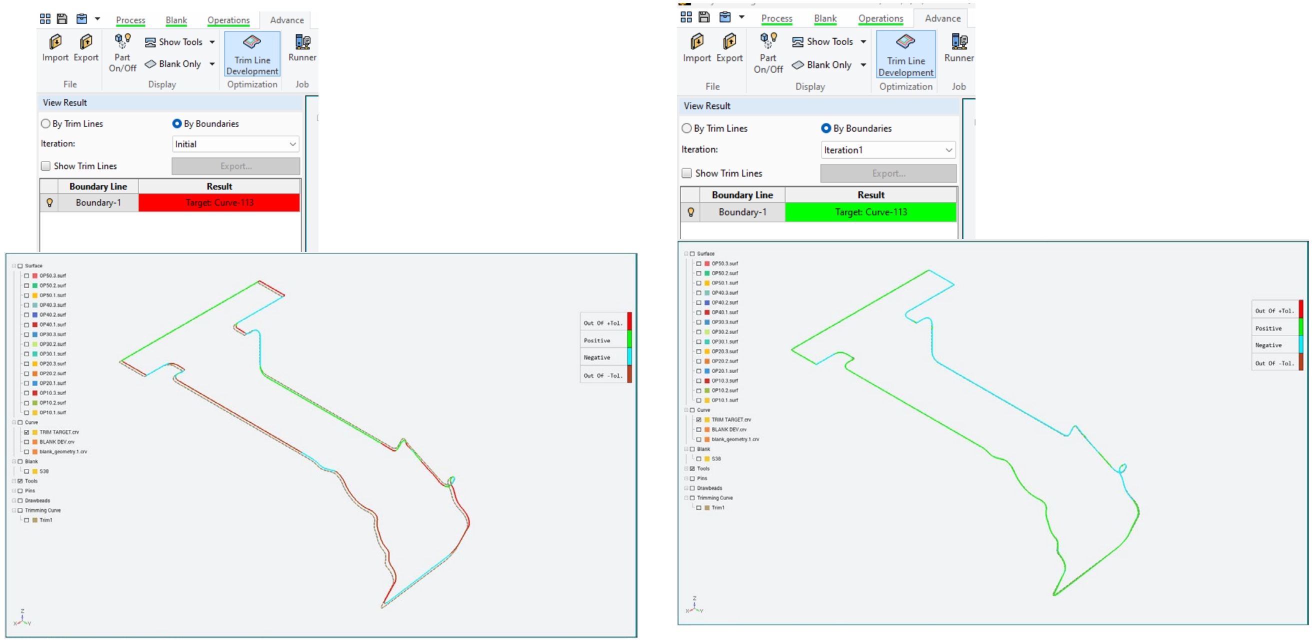
Edit the development of each trimming curve

Define target geometry

Define tolerance

Enable autoshift

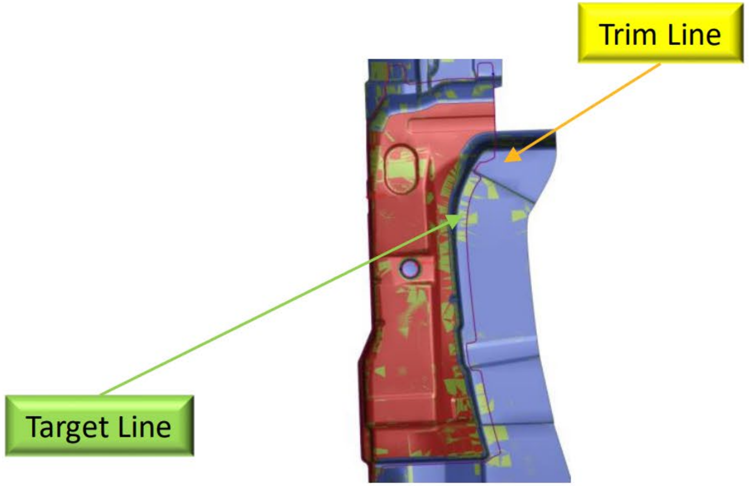
Evaluate results

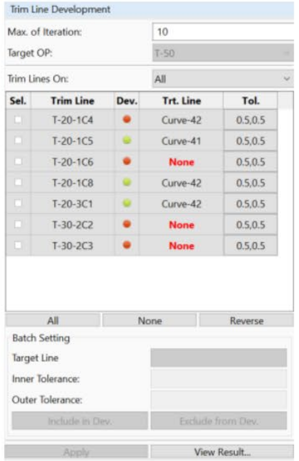


Dedicated evaluation for the development of the trimming curves

Trimming Curve Development - 2

- Example (





Trim Line Development

Max. of Iteration: 10
Target OP: T-50
Trim Lines On: All

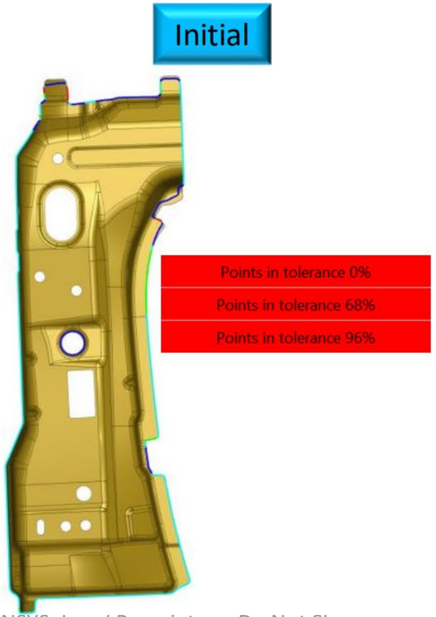
Sel.	Trim Line	Dev.	Trt. Line	Tol.
	T-20-1C4		Curve-42	0.5,0.5
	T-20-1C5		Curve-41	0.5,0.5
	T-20-1C6		None	0.5,0.5
	T-20-1C8		Curve-42	0.5,0.5
	T-20-3C1		Curve-42	0.5,0.5
	T-30-2C2		None	0.5,0.5
	T-30-2C3		None	0.5,0.5

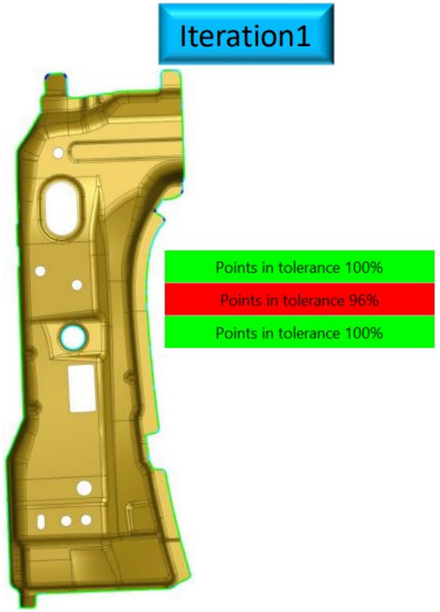
All | None | Reverse

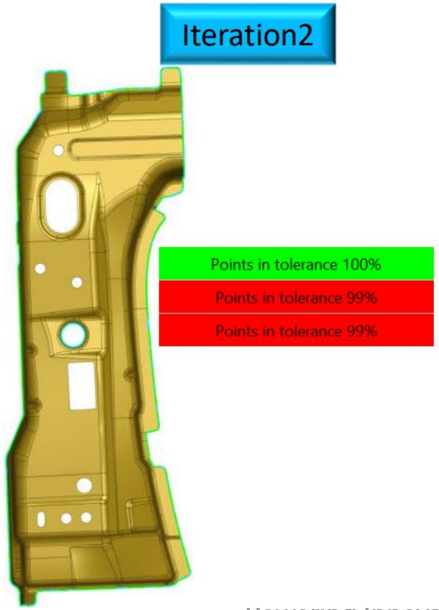
Batch Setting
Target Line
Inner Tolerance:
Outer Tolerance:
Include in Dev. | Exclude from Dev.

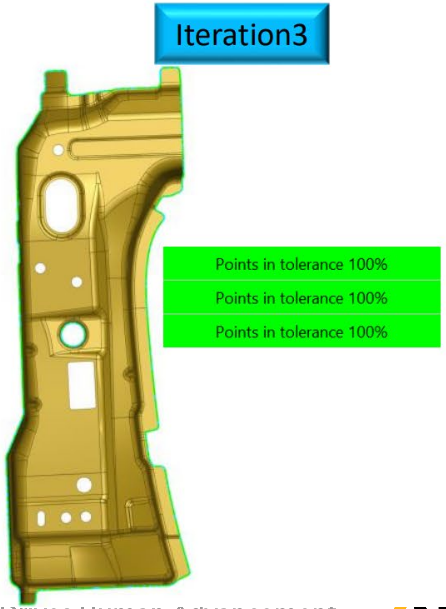
Apply | View Result...

```
$------------------------------------------
*INTERFACE_TRIMMING_CURVE_DEVELOPMENT
$  IOPTION    GAPMIN
         2       1.0
$       vx        vy        vz     depth      tol1      tolo      DMAX
       0.0       0.0      -1.0       0.0      0.50      0.50        10
```



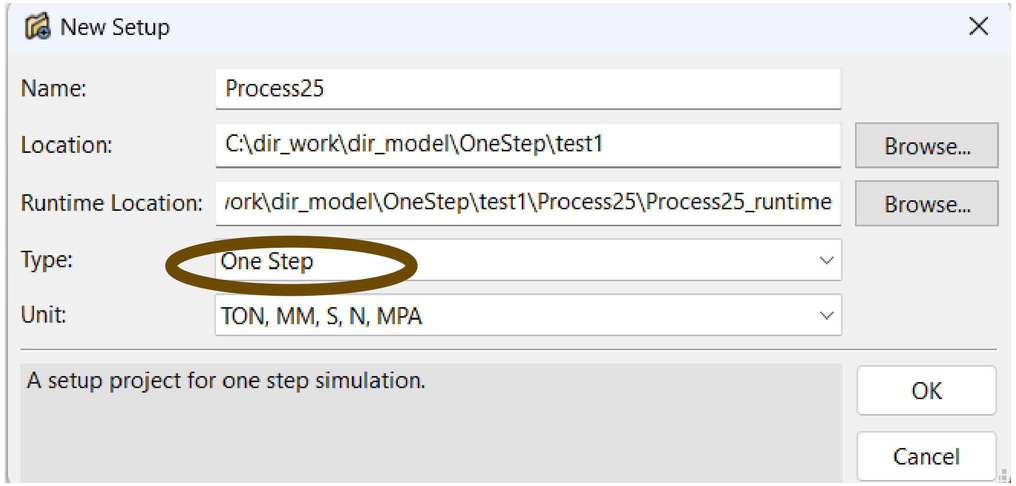


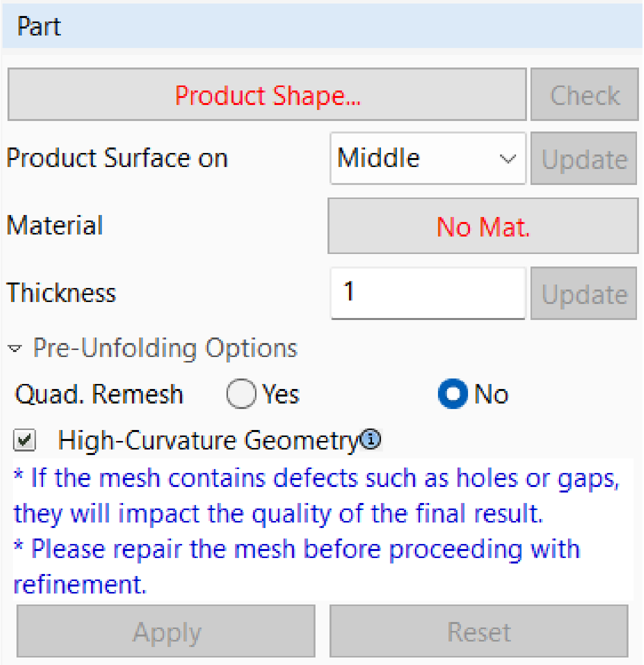




One Step Method

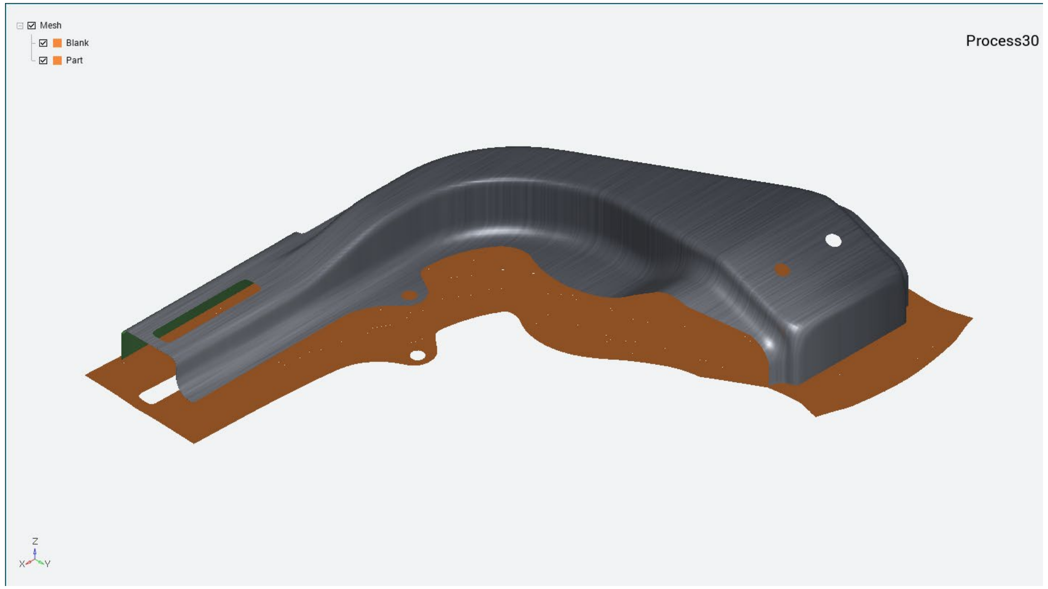
- One step method has been available in the solver
  - Several new important functions have been added
- Friendly use interface will be in the next release

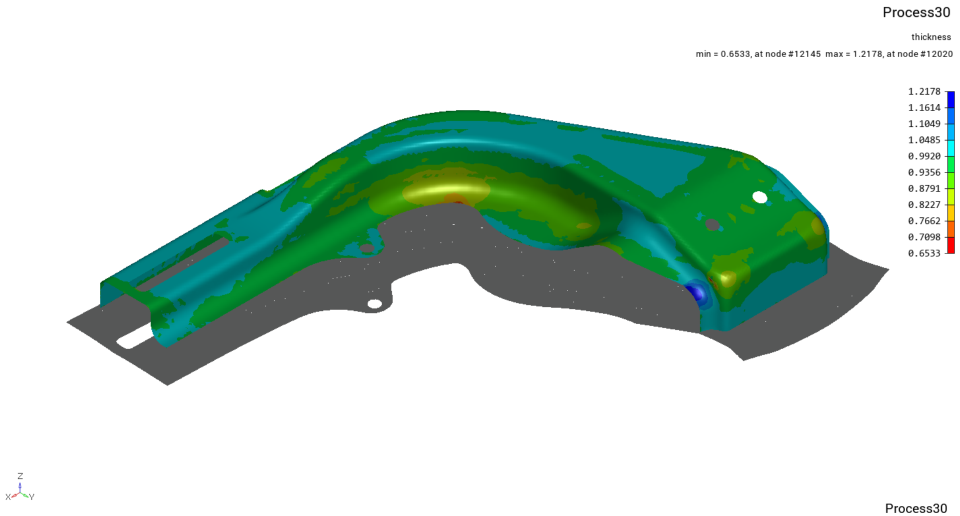


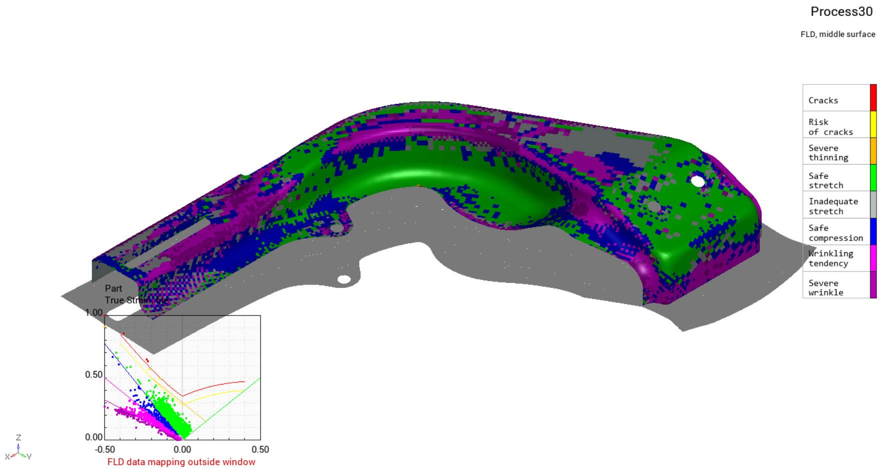


One Step Method - 2

- Auto positioning of the unfolded sheet
- Post processing: FLD, thinning…

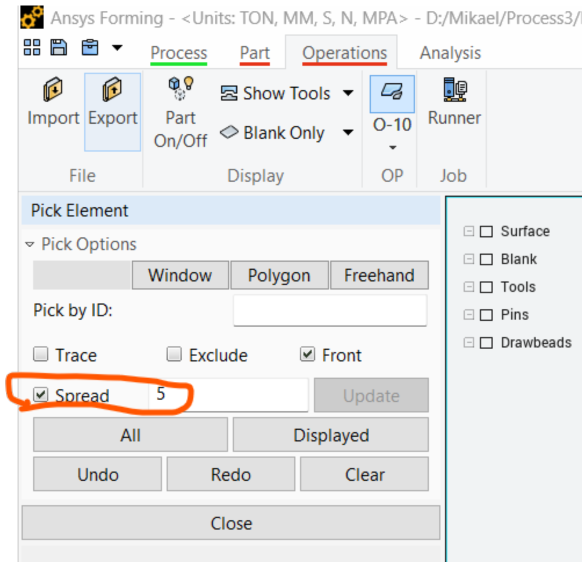


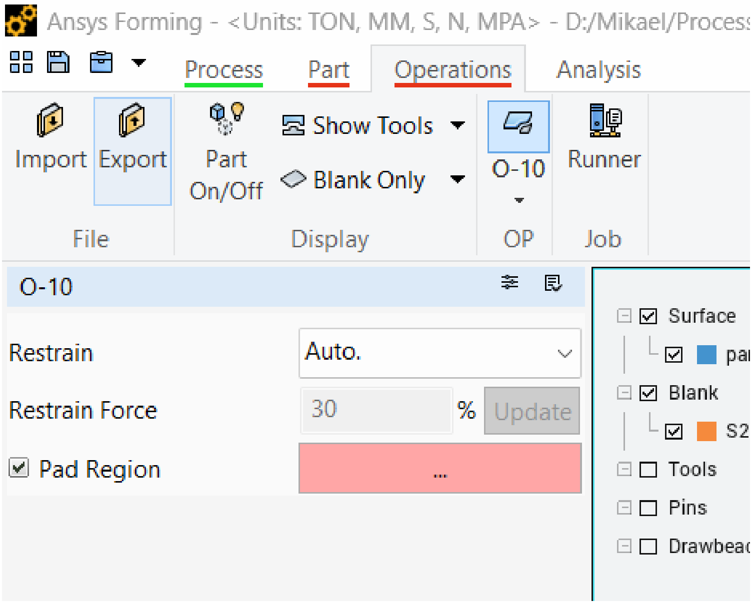


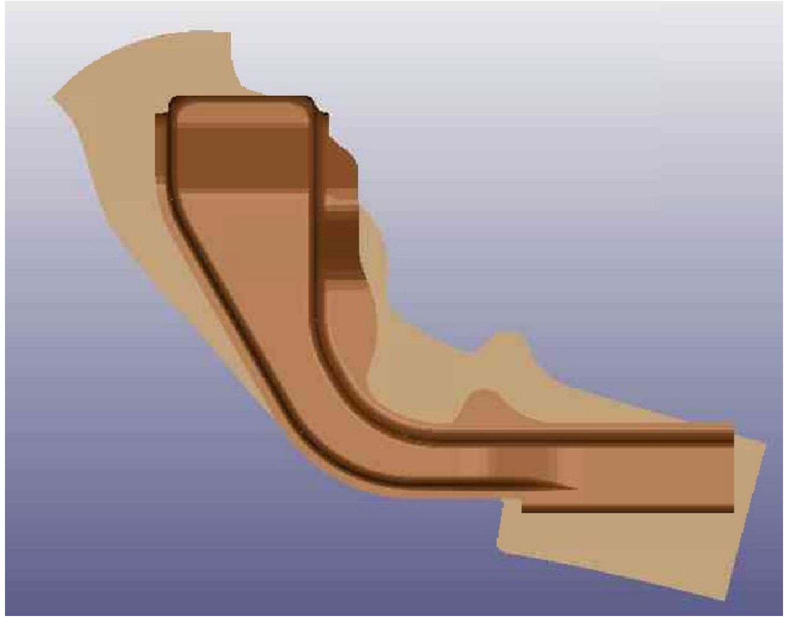


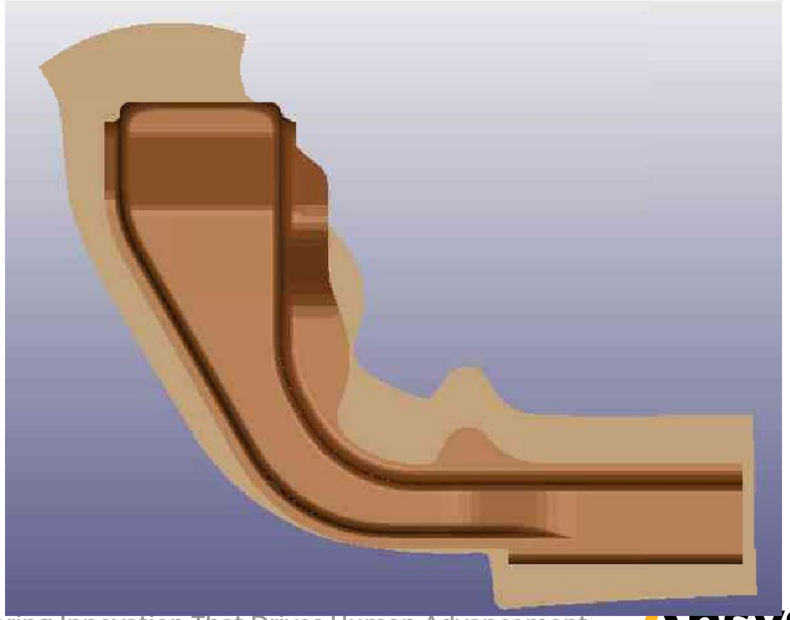
One Step Method - 3

- Pad definition
  - Material in the pad region experience small deformation
  - Pad will prevent blank from sliding over the plane





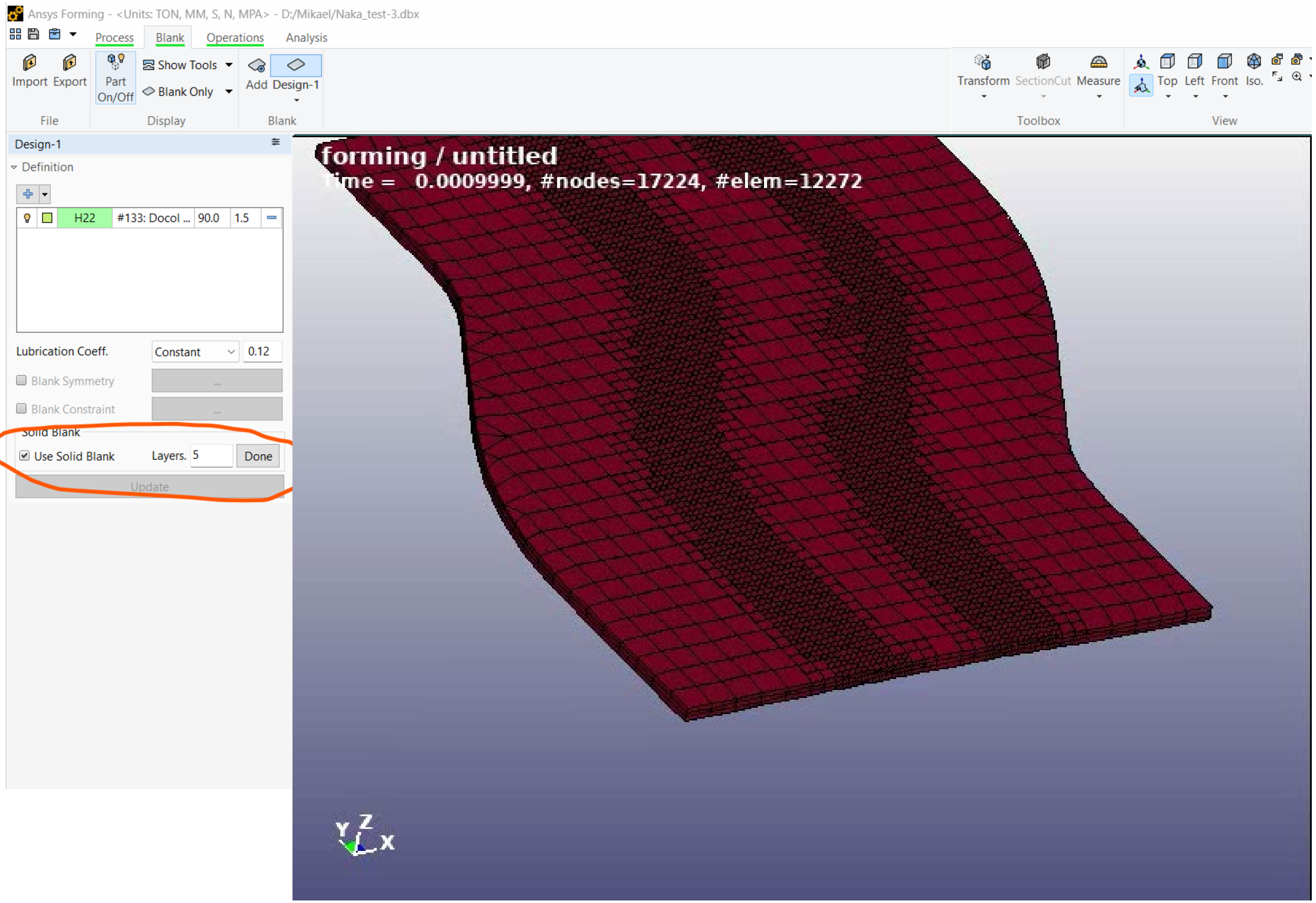




Key Functions

Solid Element

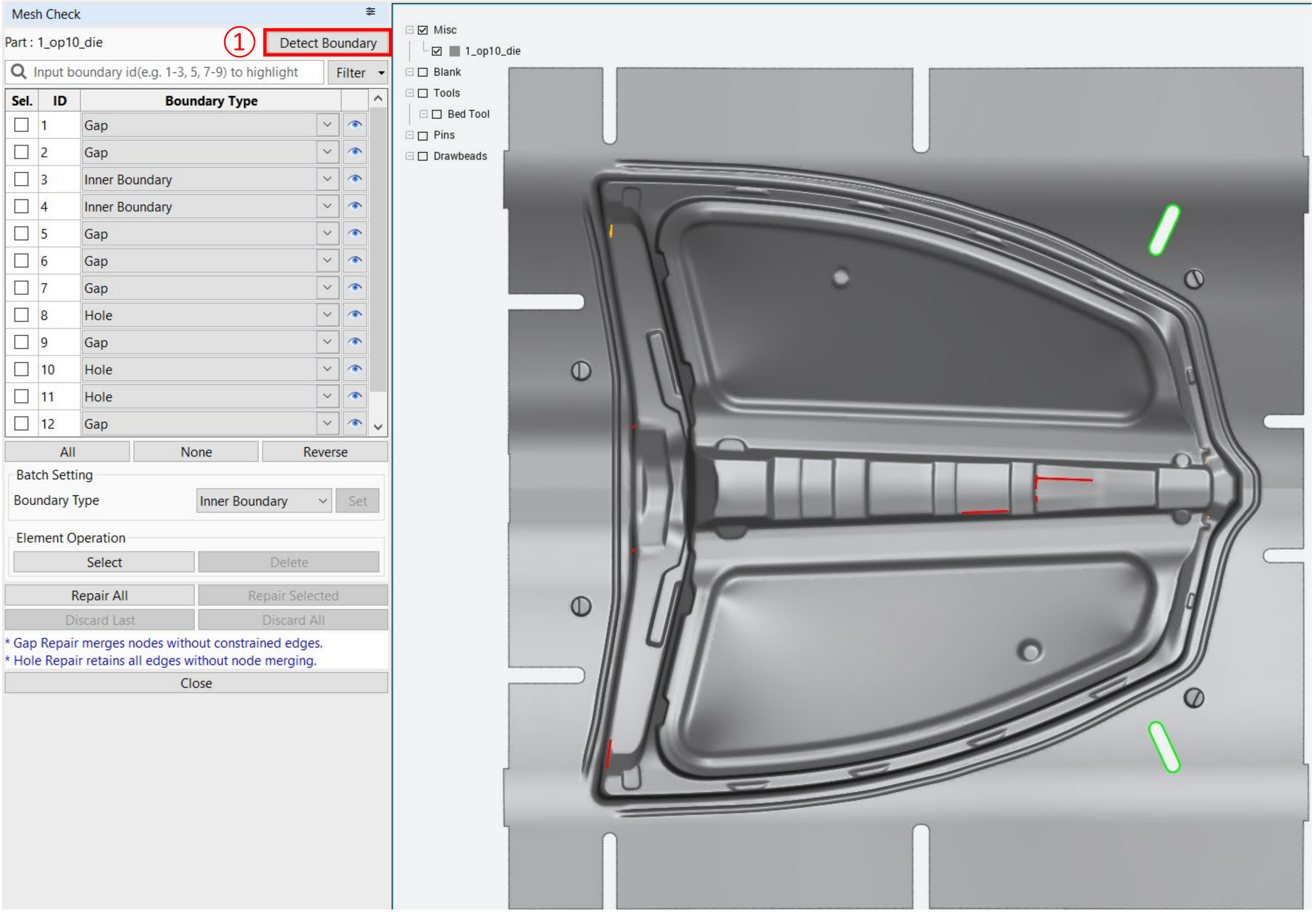
- Use only need to input the number of layers to use solid element in the preprocessing
- Some important functions in using solid elements
  - Enable mesh refinement
  - Enable contact auto normal check to avoid mesh check for the rigid tools
  - Enable contact auto move
  - All the post-processing functions are enabled



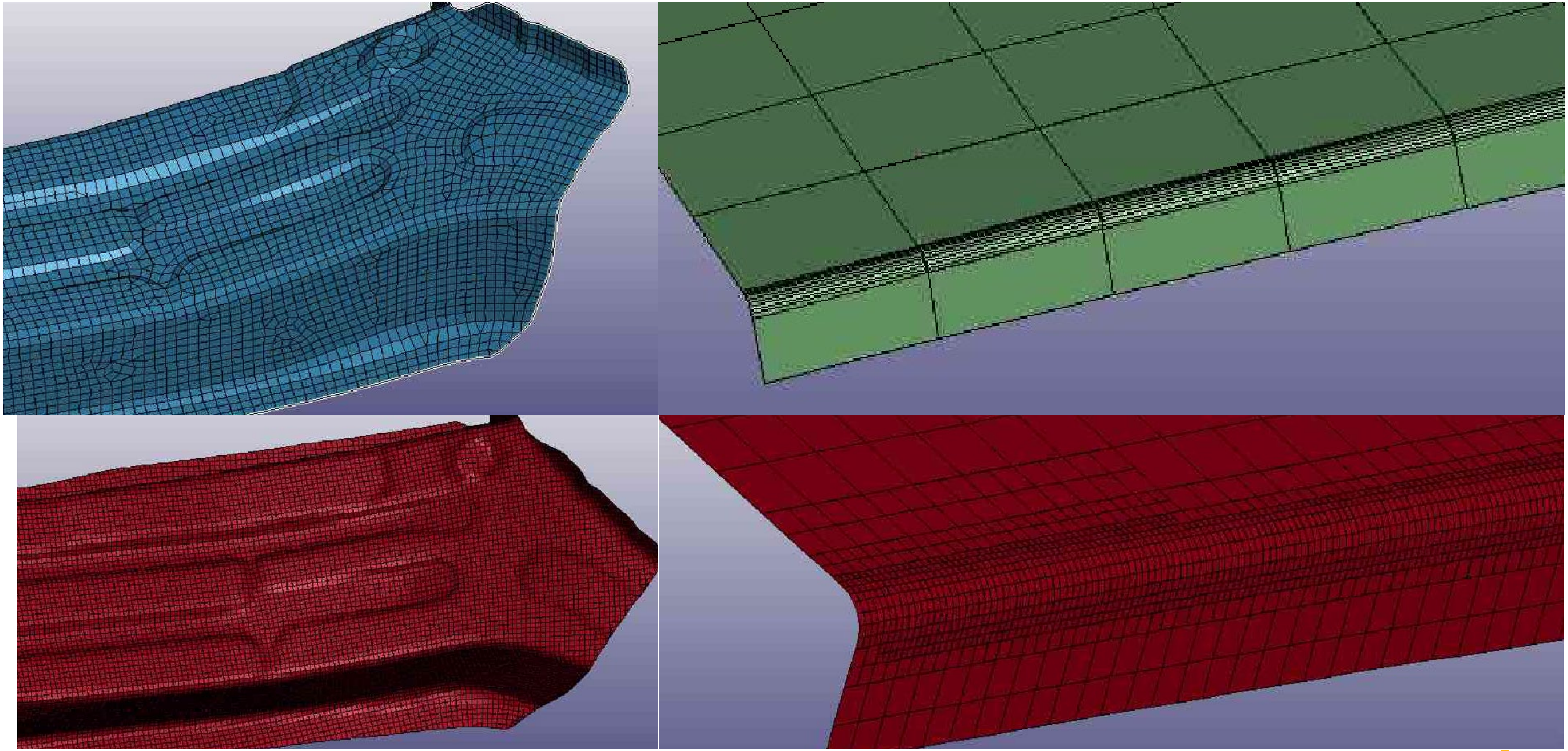
Mesh Check/Repair

- **Defects** that arise from CAD models or the mesh generation process can significantly impact the accuracy and efficiency of simulations:
  - Holes/Gaps
  - Intersecting/Overlapping/Folding elements
  - Tiny elements with poor quality
- Addressing these issues is critical to ensure the success of simulations and prevent convergence difficulties or simulation failures
- Ansys Forming's **Mesh Check** is designed to identify and rectify these defects

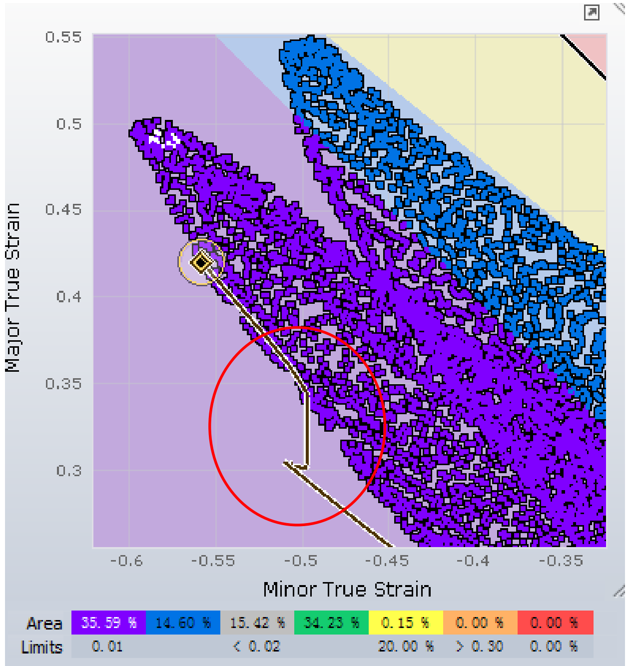
- Detect Boundary
  - *Removes* folding/tiny elements
  - *Detects* all boundaries
  - *Classifies* detected boundaries
    - **Inner Boundary**: Retained to represent the true geometry
    - **Hole**: To be repaired by the Hole-Repair method
    - **Gap**: To be repaired by the Gap-Repair method
    - Boundary Type can be changed as needed



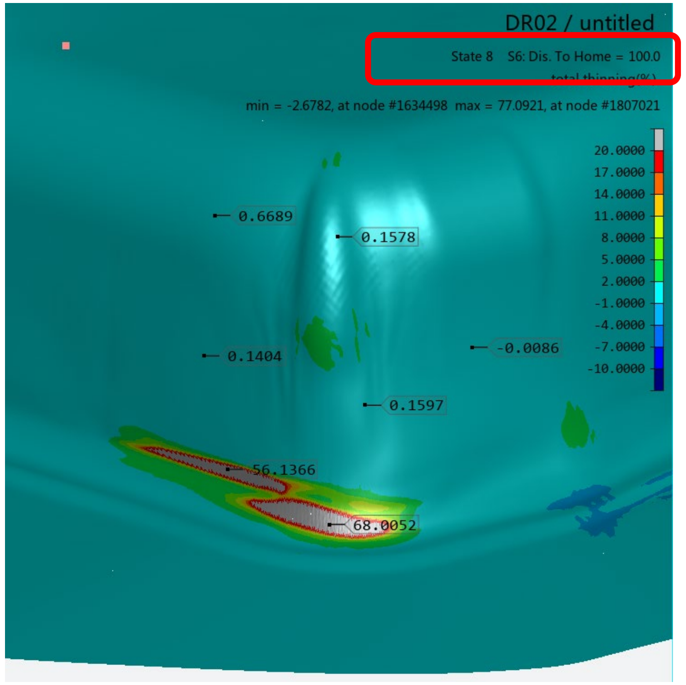
Mesh Regeneration with Mesh Adaptivity



Formability Index



Non-linear strain path

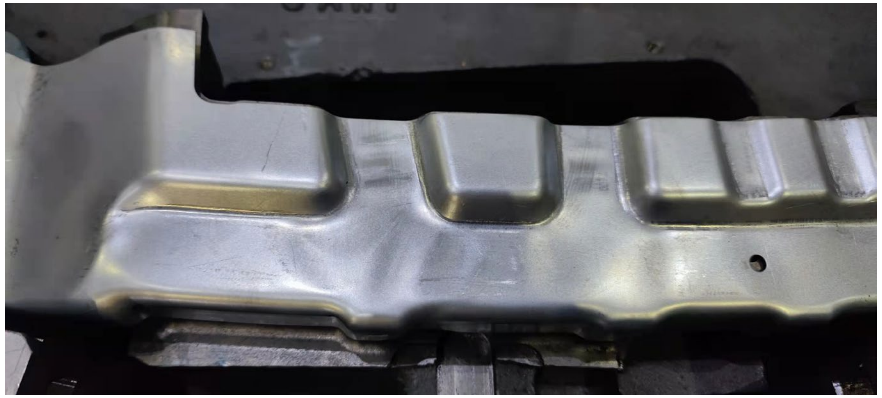


100mm to home (simulation)

100mm to home (real part）

By using Formability Index， the predicted failure is very close to reality, while fails 100mm before the die reach its home position.
With conventional FLD, the failure can not be predicted.

Wrinkling predictions



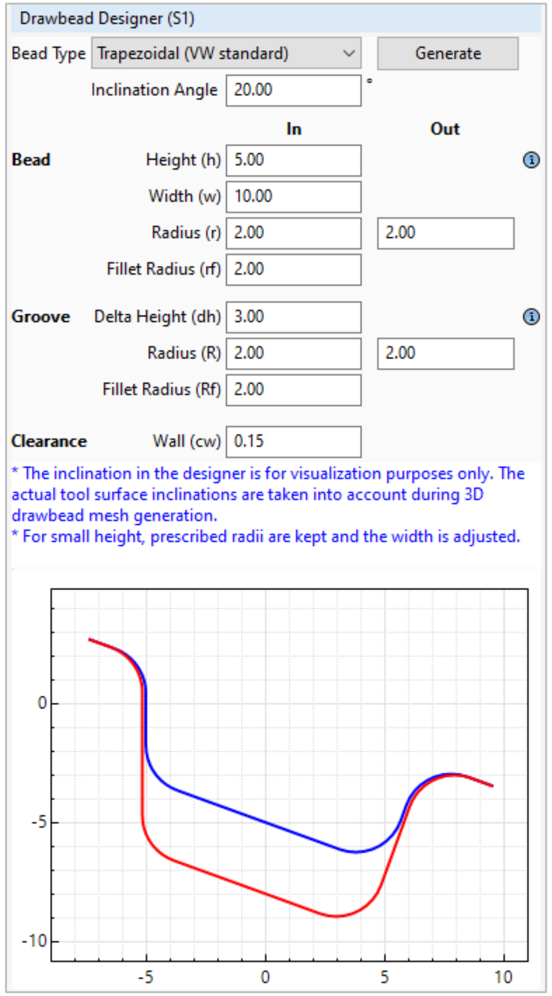
Real Part

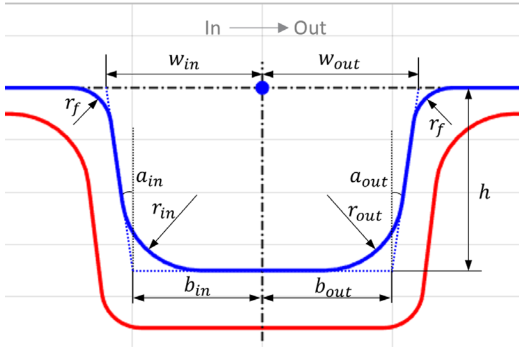
Ansys Forming prediction

Winkling is a very serious problem, especially for ultra-high strength steel
The accurate prediction of wrinkling is not easy.

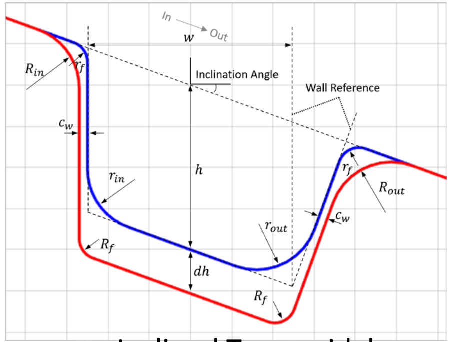
Drawbead Profile Generator

- Three new inclination-sensitive drawbead profile templates, tailored for curved surfaces

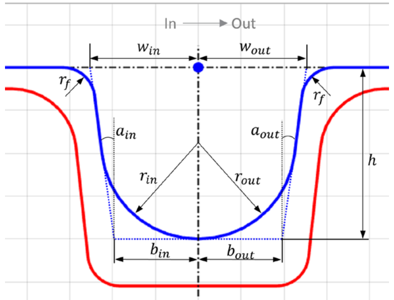




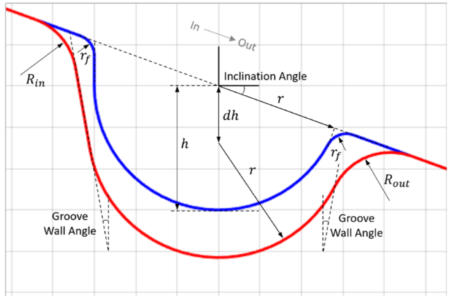
Standard Trapezoidal



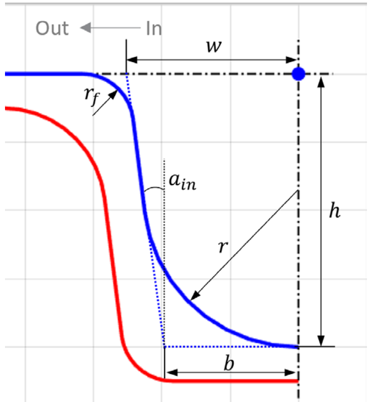
Inclined Trapezoidal



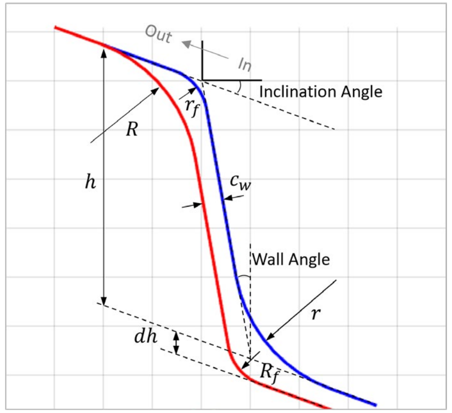
Standard Round



Inclined Round

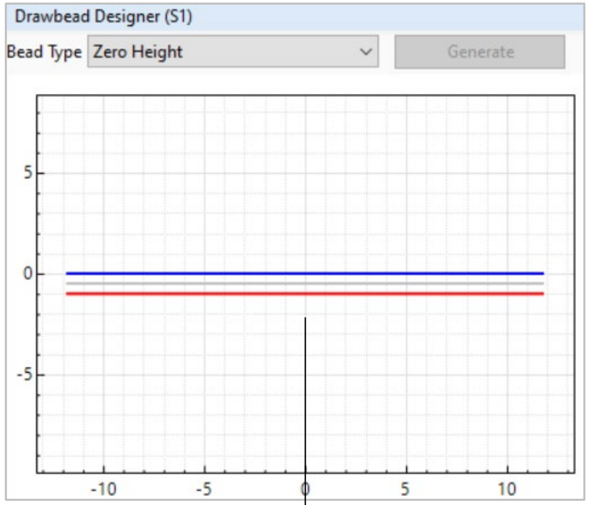


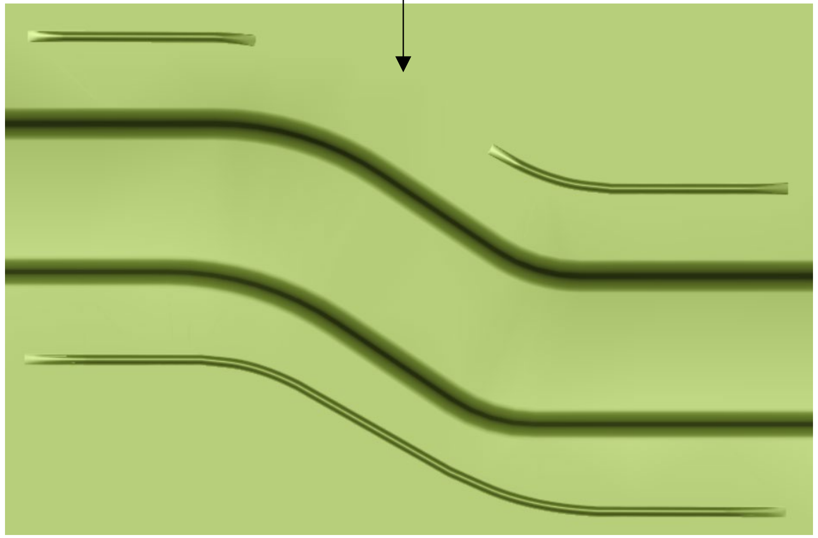
Standard Step



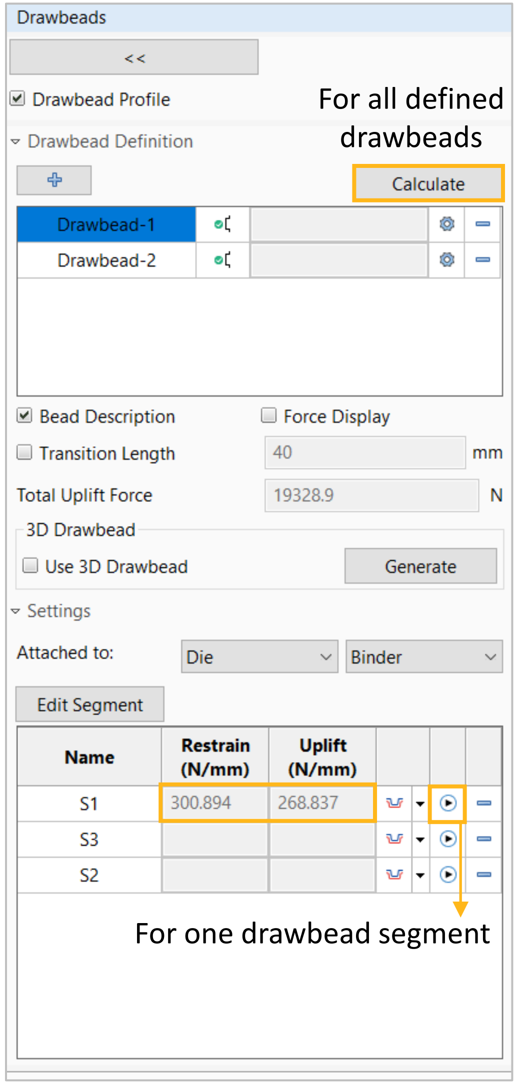
Inclined Step

- Zero-Height bead for enhanced design flexibility

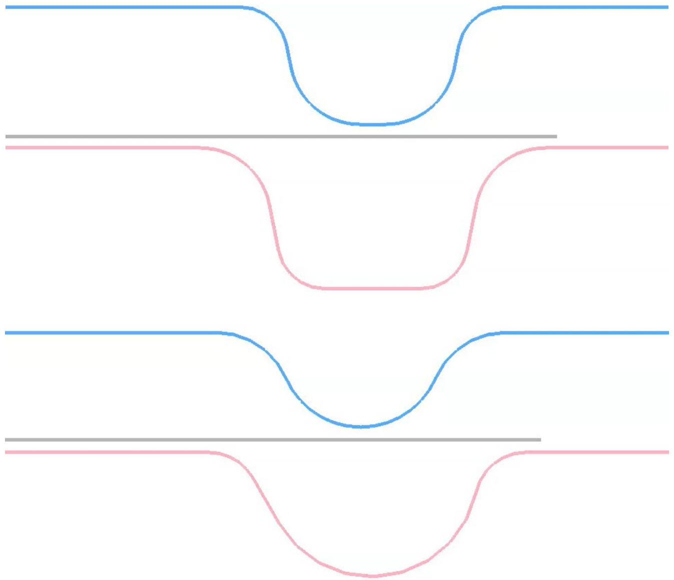


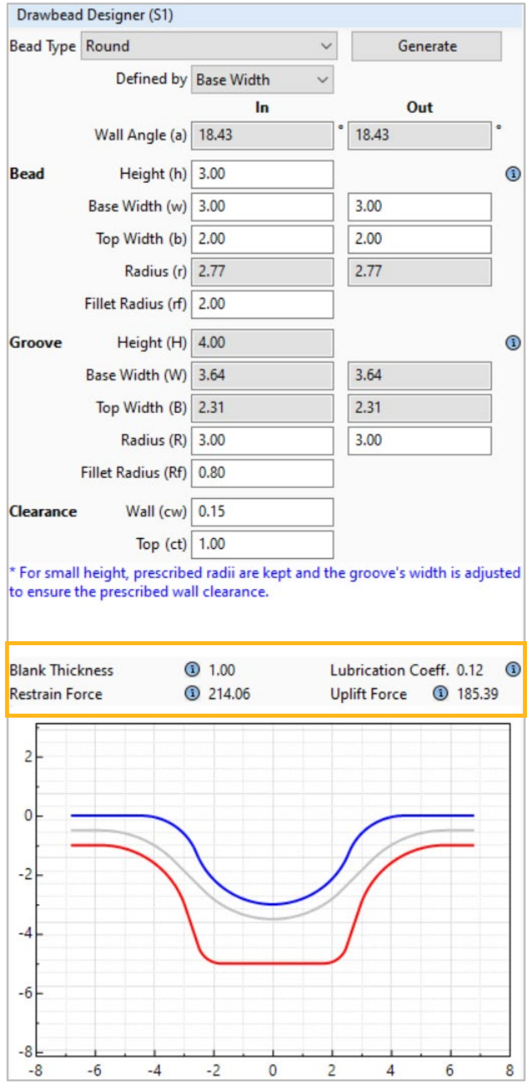


Drawbead Force Calculation and Prediction



- Fast and accurate drawbead force calculation based on drawing simulations
- Support all bead types
- Support SMP and MPP solvers
- Support constant friction, variable friction, and user-defined table friction
- Support all materials

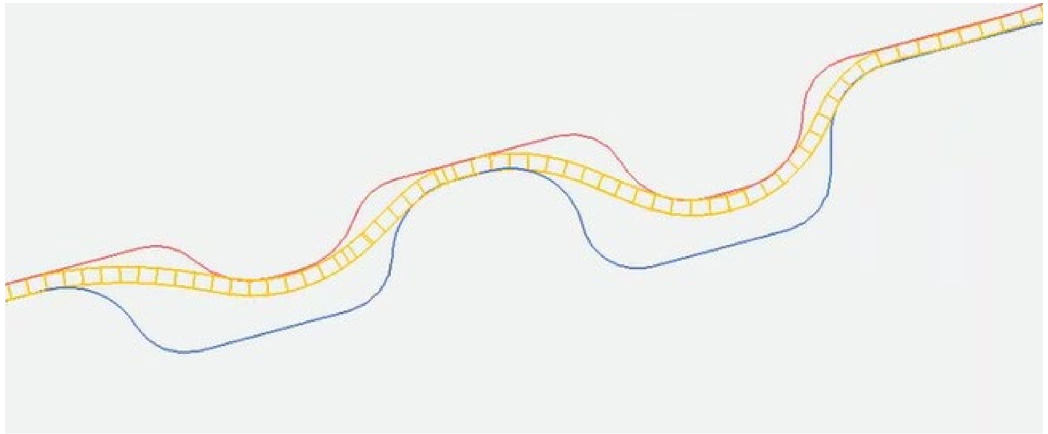




- Real-time machine learning prediction of drawbead forces
- Comparable to drawbead forces calculated by MPP solver
- Support profiles within specified parameter ranges
- Support materials with Hill’48 yield functions
- Support various blank thickness and constant friction coefficients

Non-Linear Contact

- Constant contact stiffness is commonly used in simulation
  - Large stiffness can cause contact instability
  - Small stiffness can increase penetration
- Common observation:
  - Obvious penetration is observed in curved region
- New approach
  - In smooth region, smaller contact stiffness is used
  - In curved region, large contact stiffness is used



A Universal Material Model

- The purpose of this material model is to improve the performance of the plasticity models
- The application of this material model is intended for sheet metal forming applications
- Only isotropic hardening has been supported

Default Settings

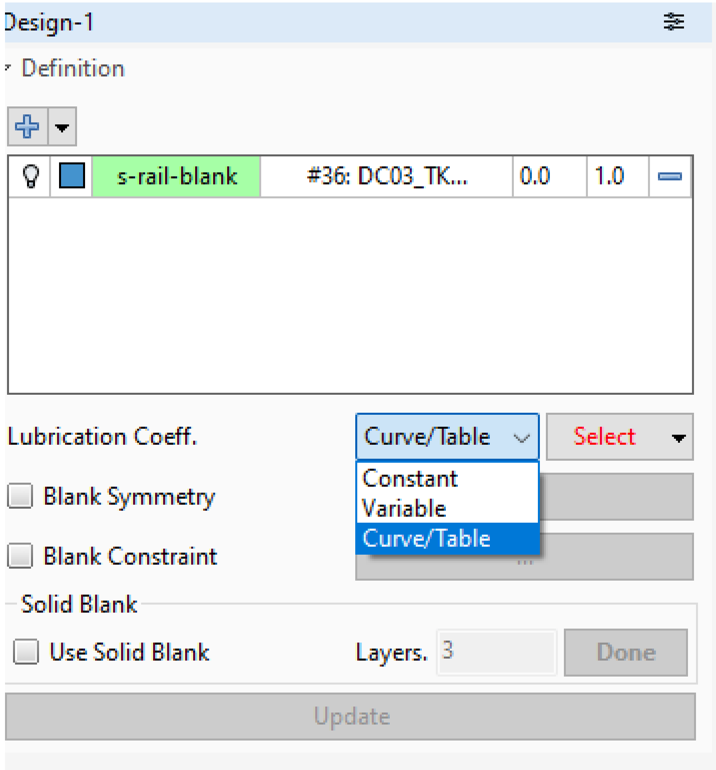
Options

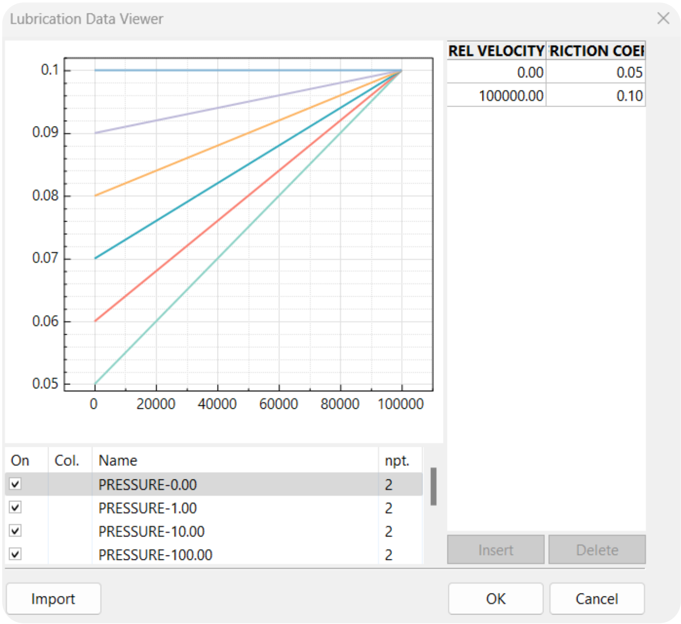
Contact Friction	0.12
☐ Contact Stiffness	0.1
☐ Shell Type (Exp.)	2
☐ Shell Type (Imp.)	16
Solid Type (Exp.)	-1
Solid Type (Imp.)	-1
☐ NIP	3
☐ Shell Shear Stiffness	0.8333
☐ Output Distance Interval	5
☐ Output Positions	5,4,3,2,1
☑ MAT320 Parameters(IPHI,ITHETA)	501,251
☐ Output Contact Info.	0

Material used	**Run time**	**Saving (%)**
3 parameter Barlat	4 hours 57 minutes 36 seconds	
3 parameter Barlat NLP	5 hours7 minutes 14 seconds	
Barlat 2000	4 hours 35 minutes 2 seconds	
MAT320 + 3 parameter Barlat	3 hours 56 minutes 9 seconds	1 hr 1min 27s (20%)
MAT320 + 3 parameter Barlat NLP	3 hours 58 minutes 16 seconds	1 hr 8min 58s (22%)
MAT320 + Barlat 2000	3 hours 56 minutes 3 seconds	0 hr 38min 59s (14%)

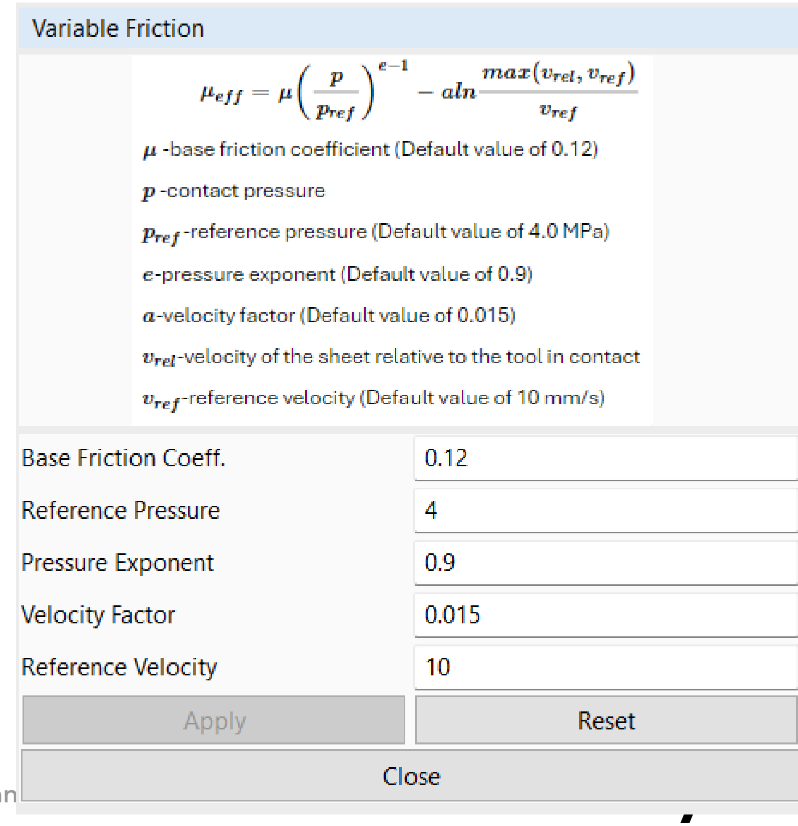
Variable Friction Coefficient

- Friction coefficient can be defined as a function of contact pressure and relative velocity
  - It support both a friction function or user input table



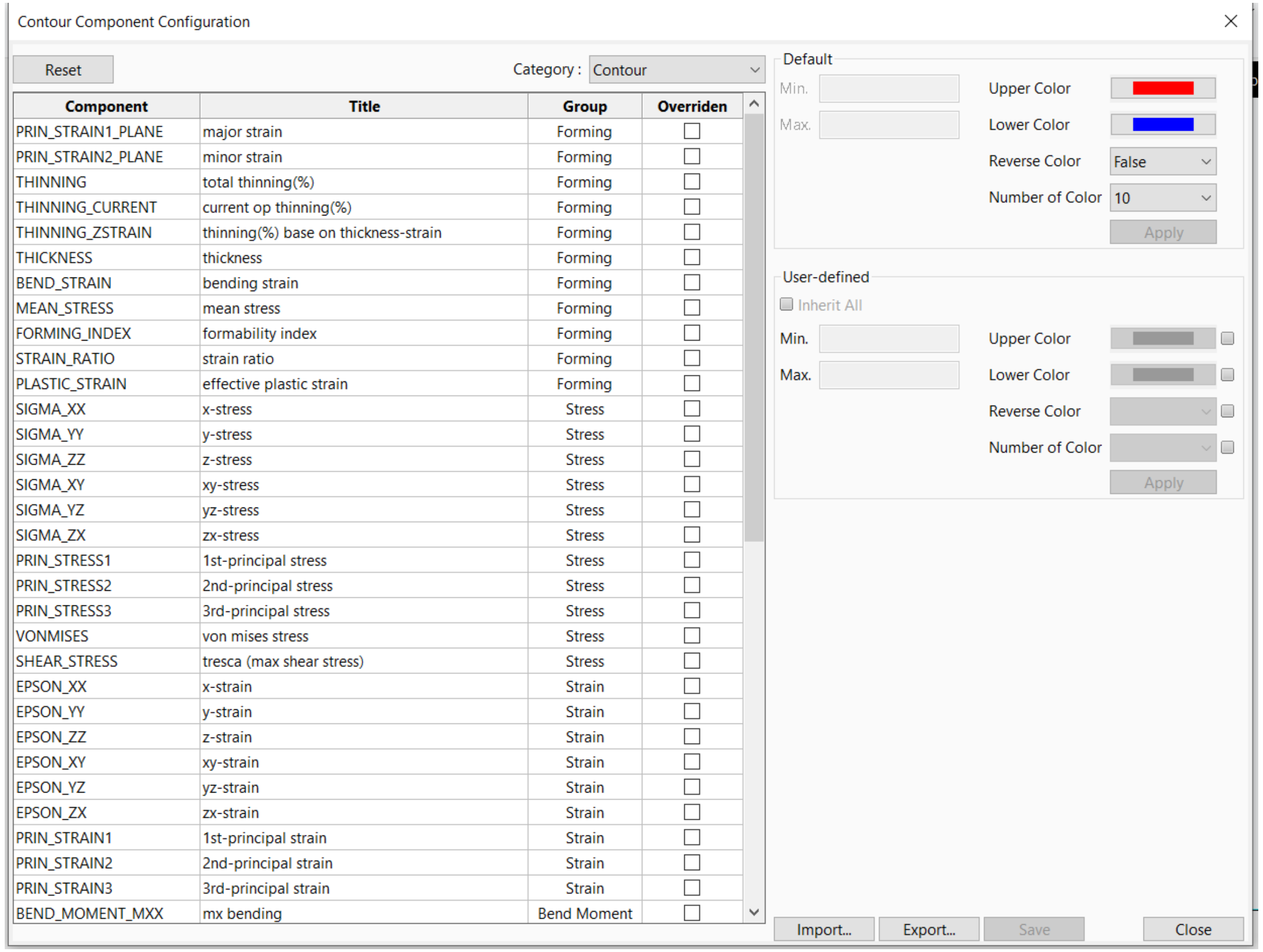


Visualization of the tabular data



Reconfigurable setting

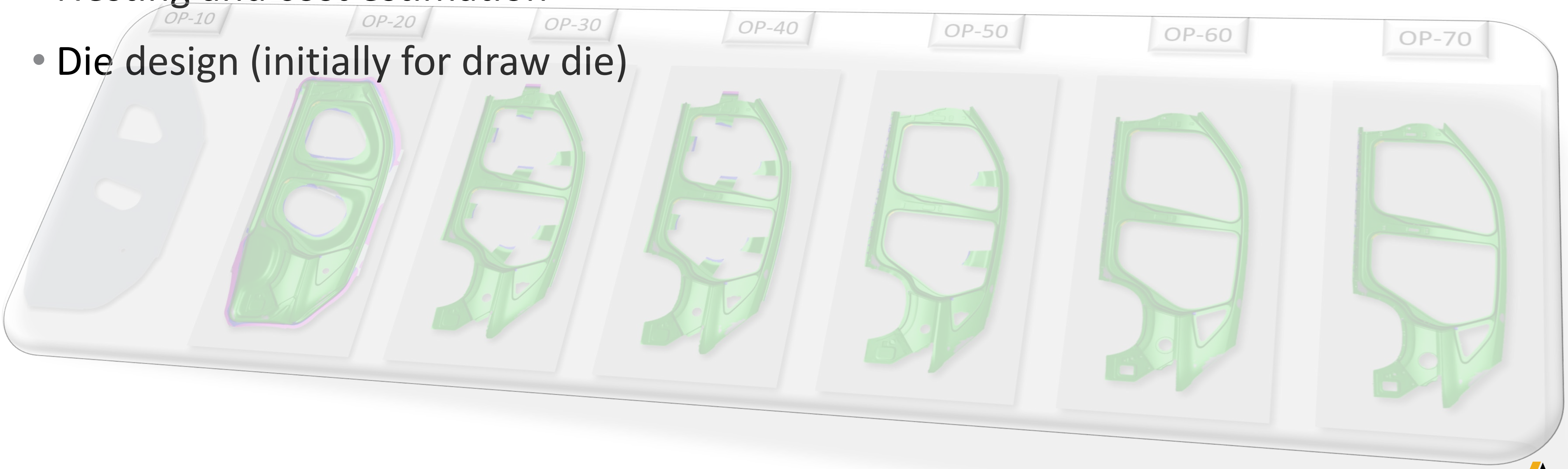
- Each company/engineer can define the setting
  - Color
  - Range
  - Easy to prepare report



Component	Title	Group	Overriden
PRIN_STRAIN1_PLANE	major strain	Forming	
PRIN_STRAIN2_PLANE	minor strain	Forming	
THINNING	total thinning(%)	Forming	
THINNING_CURRENT	current op thinning(%)	Forming	
THINNING_ZSTRAIN	thinning(%) base on thickness-strain	Forming	
THICKNESS	thickness	Forming	
BEND_STRAIN	bending strain	Forming	
MEAN_STRESS	mean stress	Forming	
FORMING_INDEX	formability index	Forming	
STRAIN_RATIO	strain ratio	Forming	
PLASTIC_STRAIN	effective plastic strain	Forming	
SIGMA_XX	x-stress	Stress	
SIGMA_YY	y-stress	Stress	
SIGMA_ZZ	z-stress	Stress	
SIGMA_XY	xy-stress	Stress	
SIGMA_YZ	yz-stress	Stress	
SIGMA_ZX	zx-stress	Stress	
PRIN_STRESS1	1st-principal stress	Stress	
PRIN_STRESS2	2nd-principal stress	Stress	
PRIN_STRESS3	3rd-principal stress	Stress	
VONMISES	von mises stress	Stress	
SHEAR_STRESS	tresca (max shear stress)	Stress	
EPSON_XX	x-strain	Strain	
EPSON_YY	y-strain	Strain	
EPSON_ZZ	z-strain	Strain	
EPSON_XY	xy-strain	Strain	
EPSON_YZ	yz-strain	Strain	
EPSON_ZX	zx-strain	Strain	
PRIN_STRAIN1	1st-principal strain	Strain	
PRIN_STRAIN2	2nd-principal strain	Strain	
PRIN_STRAIN3	3rd-principal strain	Strain	
BEND_MOMENT_MXX	mx bending	Bend Moment	

Exciting Future

- Springback compensation of line dies
- Morphing of CAD surface after springback compensation
- Auto reporting
- Nesting and cost estimation
- Die design (initially for draw die)



Ansys