3D Drawbead Design and Mesh Regeneration in ANSYS FORMING®

Xiaolong He[1], Mouhamadou Mansour Mbow[2], Junyue Zhang[3], Yi Xiao[3], Yiquan Tang[3], Kang Shen[4], Jinglin Zheng[1], Xinhai Zhu[1]

[1]Ansys Inc., Livermore, USA
[2] Ansys France SAS, Lyon, France
[3]Ansys China, Shanghai, China
[4]Ansys Germany GmbH, Hannover, Germany

Abstract

In ANSYS FORMING, we have developed innovative techniques for predicting drawbead forces and generating 3D drawbead geometry to streamline the drawbead design process, which is crucial for controlling material flow in stamping processes. These tools include a drawbead profile generator, a simulator, and a 3D drawbead generator. The profile generator creates custom drawbead profiles based on sectional design parameters, while the simulator estimates restraint and uplift forces through a strip drawing test. The 3D drawbead generator automatically creates 3D drawbead meshes from single or multiple drawbead profiles for both closed and open beads, enabling accurate material flow estimation in 3D high-fidelity stamping simulations. In addition, we introduce an innovative surface mesh regeneration method based on adaptive quadtree refinement to address issues of distorted or skewed initial meshes. This method intelligently refines regions with complex features or high gradients that require higher resolution, while discretizing smoother areas with coarser elements, optimizing computational resources without sacrificing precision. Our approach also preserves original boundaries, ensuring the fidelity of simulation results. These advancements, validated through real-world examples, significantly enhance the design capabilities and modeling performance of ANSYS FORMING in stamping simulations.

1 Introduction

In the field of sheet metal forming, drawbeads play a crucial role in controlling material flow and ensuring the quality of stamped components. These strategically placed features on the tooling surface introduce resistance during metal flow, helping to achieve desired part shapes while preventing defects such as wrinkles and cracks. The design of 3D drawbeads is essential to optimize material flow, maintain minimal stretching, minimize springback, and ensure a consistent manufacturing process.

To enhance the drawbead design capabilities in ANSYS FORMING, we have developed a suite of advanced tools, including:

- **2D drawbead profile generator**: Produces customized drawbead profiles based on sectional design parameters.
- **Drawbead simulator**: Estimates drawbead forces corresponding to specific drawbead profiles and materials.
- **3D drawbead generator**: Creates 3D drawbead meshes based on drawbead profiles.

In addition to precise drawbead design, maintaining mesh quality is crucial for the accuracy, efficiency, and stability of simulations. Poor mesh quality, such as distorted or excessively skewed elements, can lead to numerical issues. Complex geometries often demand finer mesh resolution, while simpler areas can tolerate coarser meshes.

To address these challenges, we introduce an adaptive mesh regeneration technique for surface meshes, which enhances mesh quality while preserving the geometric accuracy. This method generates high-quality quadrilateral elements in the interior. The adaptive mesh regeneration technique allows for multi-resolution refinement along the boundary and surface, providing flexible control over mesh resolution in critical regions. This innovative method is highly beneficial for a wide range of applications, such as one-step simulations, tool wear prediction, hemming simulations, crash simulations, and much more.

2 Drawbead Design

Fig. 1 illustrates a common approach to drawbead design that involves generating appropriate drawbead geometries in CAD, meshing them, and incorporating them into forming simulations. The tool surfaces, including the drawbead design, are finely meshed, leading to a large number of elements. If the simulation results of the sheet metal, such as thinning, formability, and springback, are unsatisfactory, modifications to the drawbead geometry are necessary. This leads to a repetitive cycle of geometry adjustments, remeshing, and re-simulating, which is both time-consuming and computationally expensive. The inclusion the detailed drawbead mesh in forming simulations significantly increases computational cost. Each modification requires substantial resources for geometry processing and simulation reruns, slowing down the entire optimization and design process, making it highly inefficient.

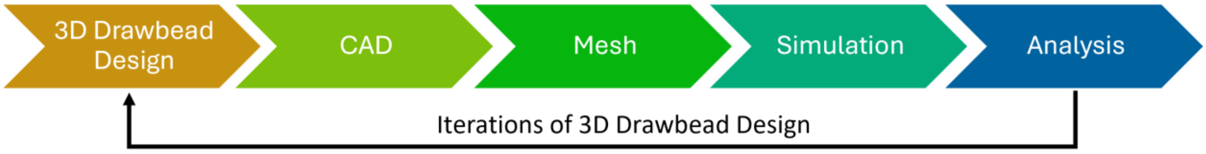


Fig.1: Drawbead design cycle with 3D geometric bead.

To improve the efficiency of the drawbead design cycle, we have developed a set of innovative tools in ANSYS FORMING, including a 2D drawbead profile generator for creating customized drawbead profiles based on sectional design parameters, a drawbead simulator for estimating restraint and uplift forces corresponding to specific drawbead profiles and materials, and a 3D drawbead generator for creating drawbead meshes based on profiles. These tools eliminate the need to include detailed 3D drawbead geometries at the beginning of the drawbead design. Instead, users can import only the tool surfaces and design 2D drawbead profiles along the drawbead line. By applying the drawbead forces estimated from the drawbead simulator along the drawbead line, users can simulate the effects of the drawbeads. If the simulation results are unsatisfactory, users can conveniently modify the 2D drawbead profile, reapply the drawbead forces, and rerun the simulation, making the design process much more efficient. Once the results meet the requirement, a high-fidelity 3D drawbead mesh can be generated based on the final 2D profile for validation. This streamlined process reduces computational costs and simplifies iterative design, allowing for faster optimization. This design cycle is illustrated in Fig. 2.

Fig.2: Drawbead design cycle with 2D profile bead and 3D geometric bead.

In ANSYS FORMING, the 2D drawbead profile generator can be activated from the Drawbeads settings panel by selecting “Drawbead Profile”, as shown in Fig. 3. Three pre-defined profile templates are available, including the Trapezoidal bead, Round bead, and Step bead. These templates offer flexibility for different design needs, allowing users to customize the drawbead profile according to specific requirements. Fig. 4 illustrates the parameterization of these profile templates, which corresponds to the user interface shown in Fig. 3. Users can define the profile by either the parameters related to the base and top width or the wall angle of the profile. Once the sectional design parameters are entered, users can generate and visualize the profile by clicking on the “Profile” button within the profile generator. This functionality simplifies the process of designing 2D drawbead profiles, enabling users to quickly customize, generate, and apply profiles in simulations.

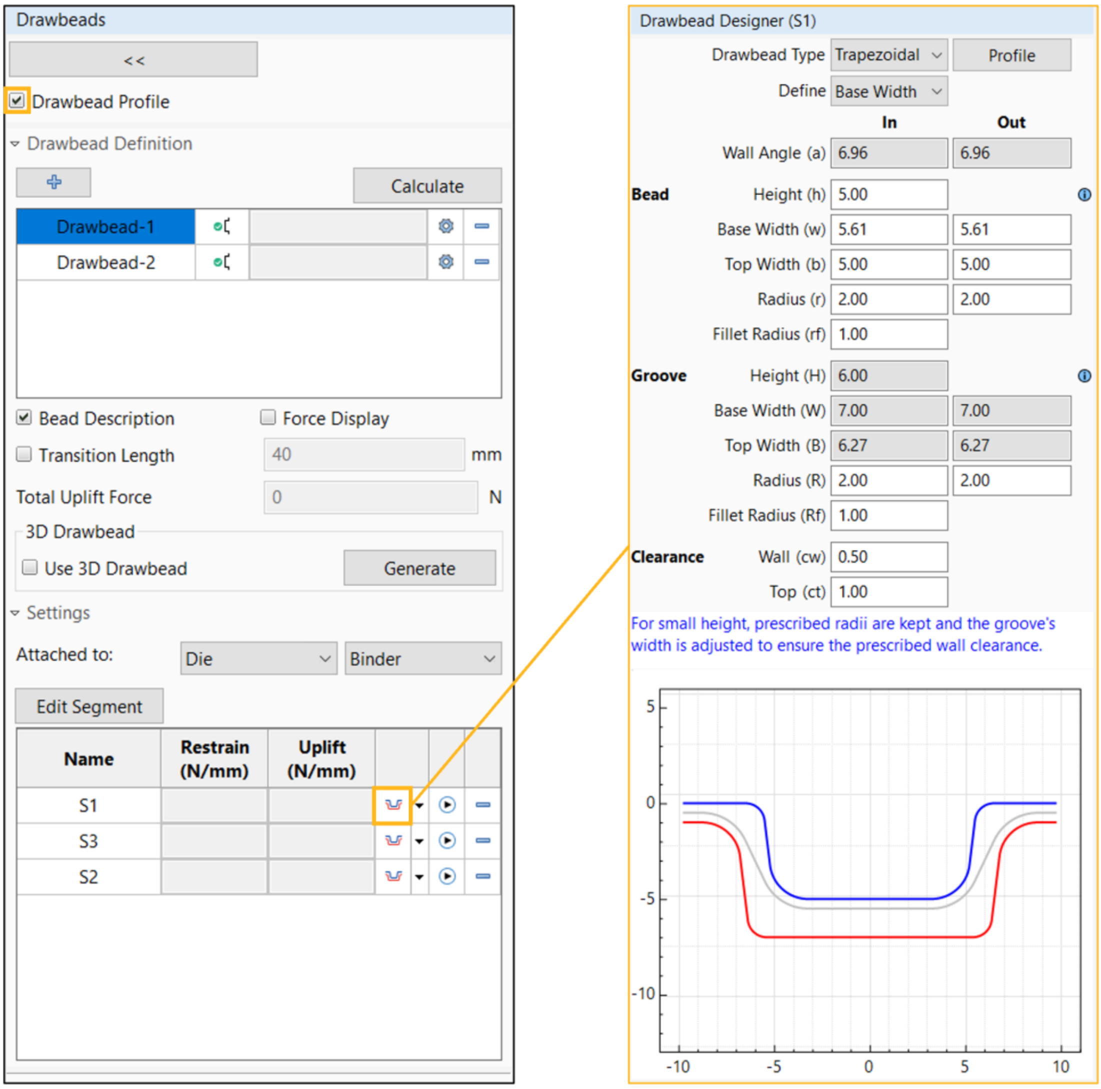
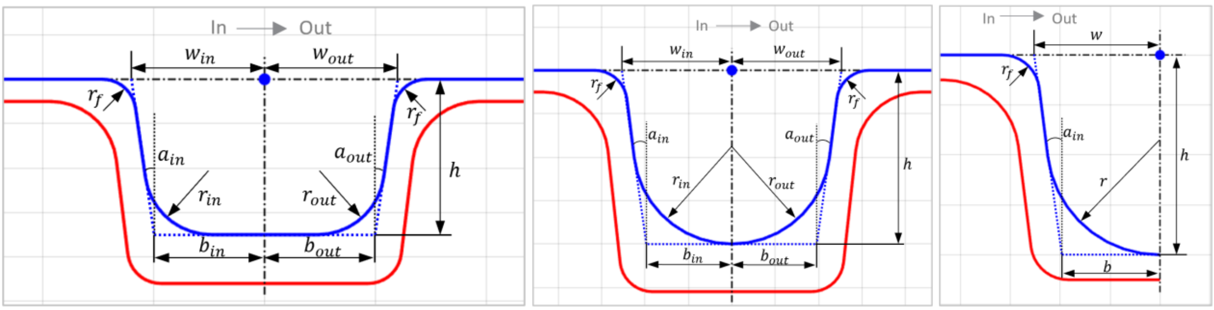


Fig.3: The user interface of the 2D drawbead profile generator.



(a) Trapezoidal bead template (b) Round bead template (c) Step bead template

Fig.4: Pre-defined drawbead profile templates: (a) trapezoidal bead; (b) round bead; (c) step bead.

3 Drawbead Force Estimation

Fig. 5 illustrates the restraint and uplift forces acting on the sheet metal (blank) as it passes through a drawbead during forming. As the blank is drawn through the bead, two primary forces are exerted:

- **Restraint Force**: This force opposes the movement of the blank, providing resistance to control material flow and preventing defects like wrinkles. The restraint force is influenced by the shape and configuration of the drawbead.

- **Uplift Force**: This vertical force lifts the blank as it passes through the bead, altering its trajectory and contributing to the material's formability.

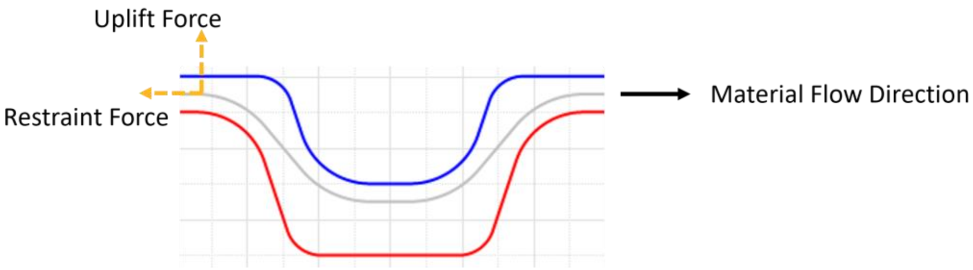


Fig.5: Schematics of restraint and uplift forces exerted on the blank through a drawbead.

Once the drawbead profile is defined, the restraint and uplift forces for a specific drawbead profile and material can be estimated by using the "Start Segment Calculation" or the "Calculate" buttons.
- "Start Segment Calculation": Computes the drawbead forces for the selected segment.
- "Calculate": Runs the calculation for all segments of drawbeads at once.

After the calculation is finished, the restraint and uplift forces are displayed in the table. To streamline the process, if multiple segments share the same drawbead profile, only one calculation is required, avoiding redundant computations. The estimated uplift forces for all segments of drawbeads are then used to update the "Total Uplift Force".

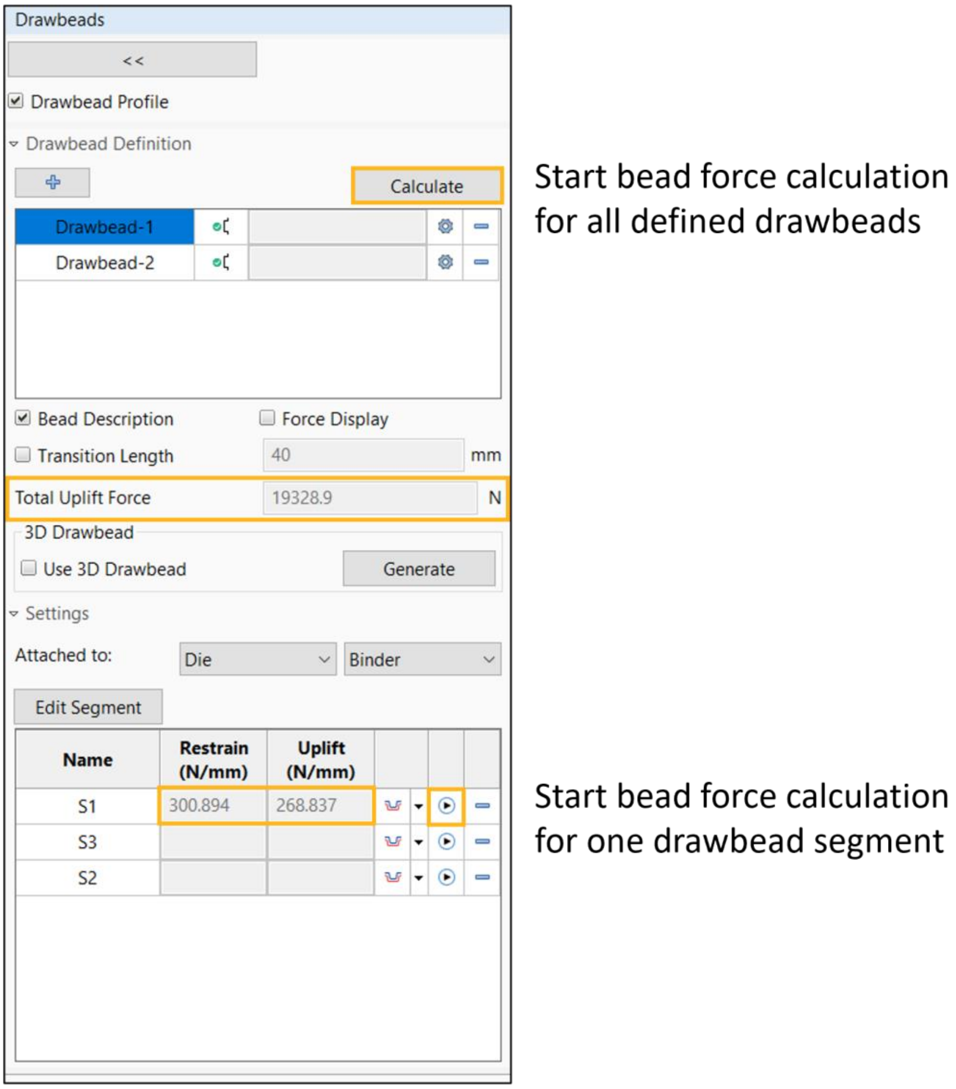


Fig.6: The user interface of drawbead settings and buttons for drawbead force estimation.

4 Profile Bead Modeling

Fig. 7 illustrates a model that features a defined profile bead along the designated drawbead lines.

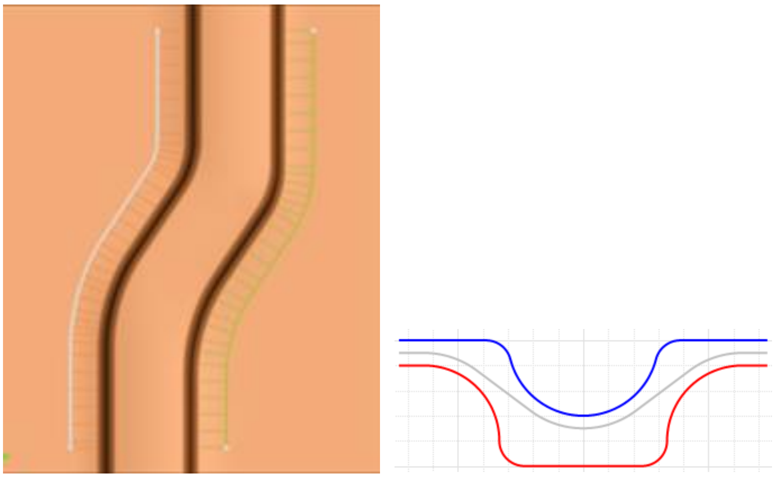
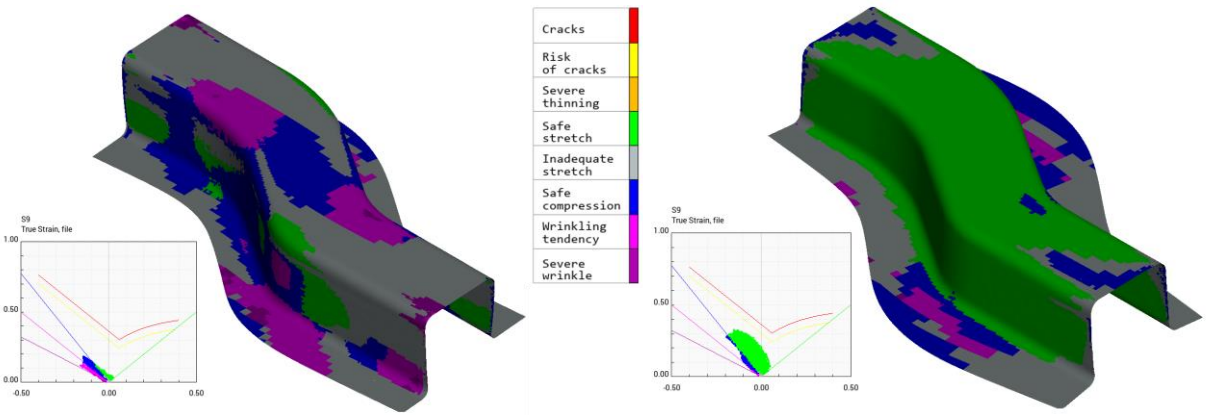


Fig.7: Model with profile bead and corresponding bead forces display.

Fig. 8 compares the forming limit diagram from a model with profile beads to that of a model without profile beads. The results clearly illustrate that the introduction of profile beads significantly reduces the wrinkling tendency and prevents severe wrinkles during the forming process. The use of profile beads improves control over material flow, ensuring a more uniform stretch of the sheet metal, which helps in utilizing the material more effectively. This example demonstrates the effectiveness of profile bead modeling in improving formability and enhancing the overall quality of the stamped part.



(a) Forming without profile bead (b) Forming with profile bead

Fig.8: The forming limit diagrams of two models: (a) without profile bead; (b) with profile bead.

5 3D Geometric Drawbead Modeling

The high-fidelity 3D drawbead mesh can be generated by clicking the “Generate” button under the “3D Drawbead” section, as shown in Fig. 9. Selecting the “Use 3D Drawbead” option enables the forming simulation to run with the 3D drawbead mesh. For each drawbead, clicking the gear icon opens the “Drawbead Settings” that allows for user customization for the selected drawbead. Figs. 10 - 13 illustrate the parameters related to the “Drawbead Settings”. The information icon next to each parameter provides explanations. The “Phase In Distance” and “Phase Out Distance” parameters represent the fading distances at the start and end of the bead, respectively, for open beads.

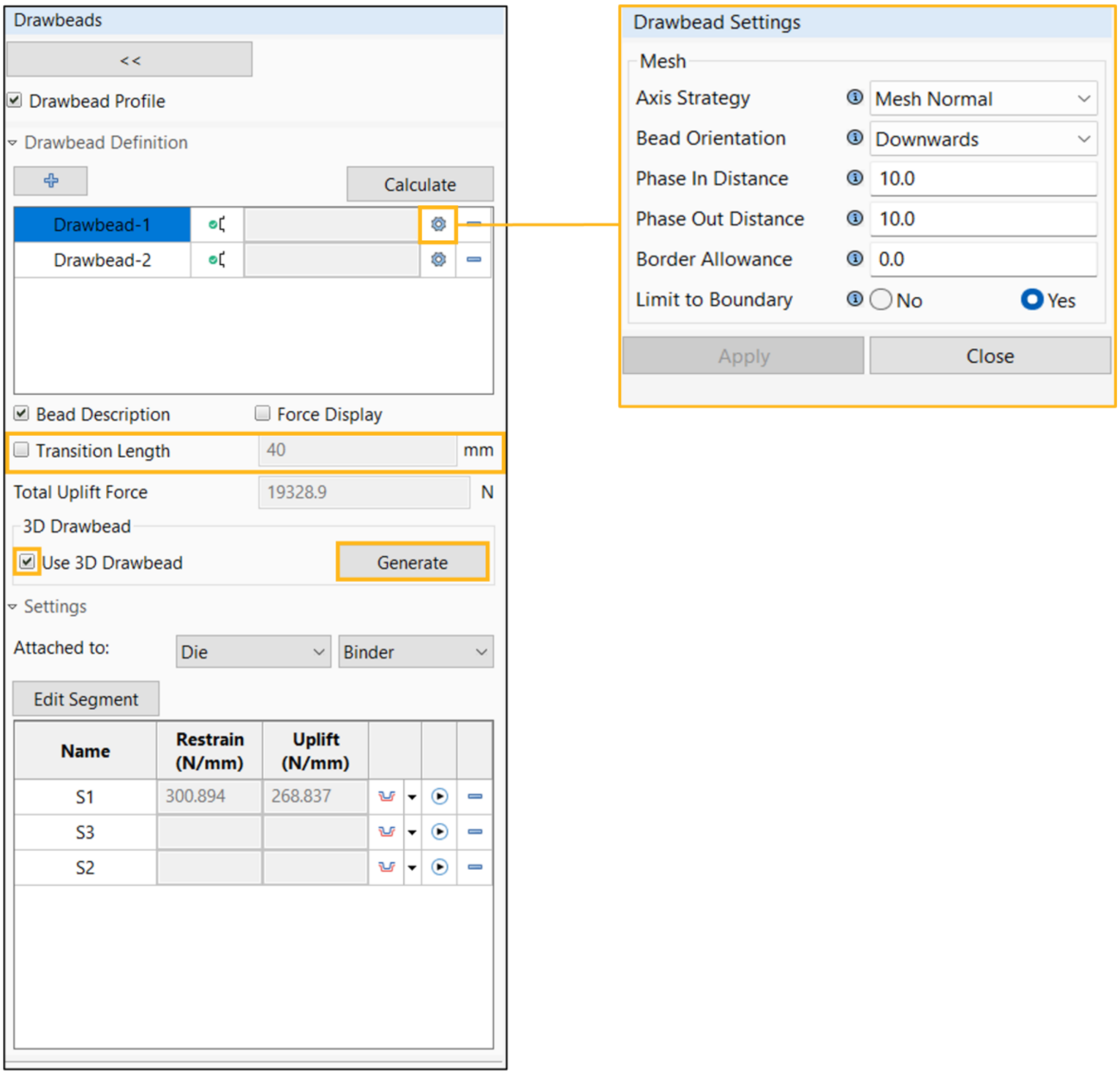
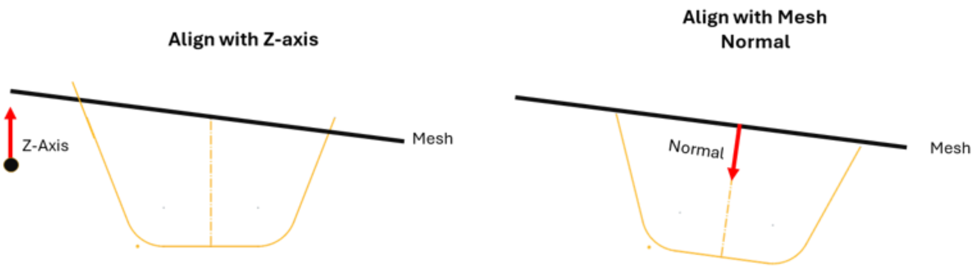


Fig.9: The user interface of 3D drawbead generator.

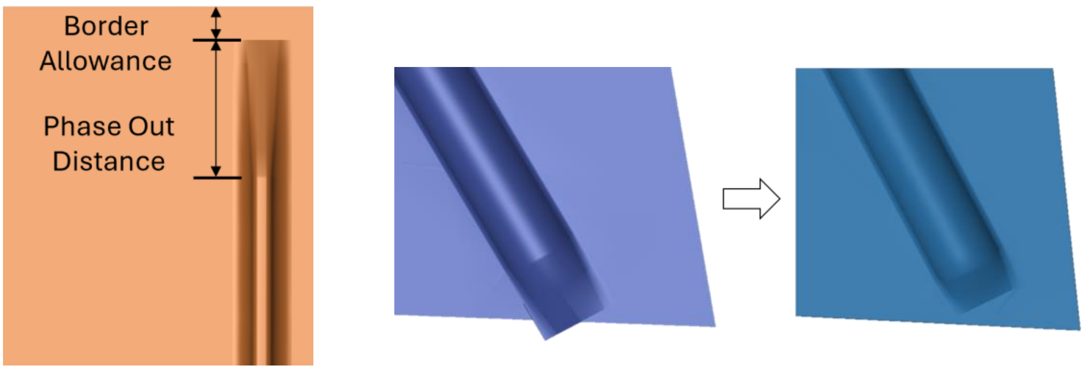


(a) Z-Axis (b) Mesh Normal

Fig.10: Schematics of two options of "Axis Strategy" in 3D drawbead settings: (a) Z-Axis; (b) Mesh Normal.

(a) Upwards (b) Downwards

Fig.11: Schematics of two options of "Bead Orientation" in 3D drawbead settings: (a) Upwards; (b) Downwards.



(a) (b) Limit to Boundary

Fig.12: Illustration of 3D drawbead settings for open beads: (a) "Border Allowance" and "Phase Out Distance"; (b) "Limit to Boundary".

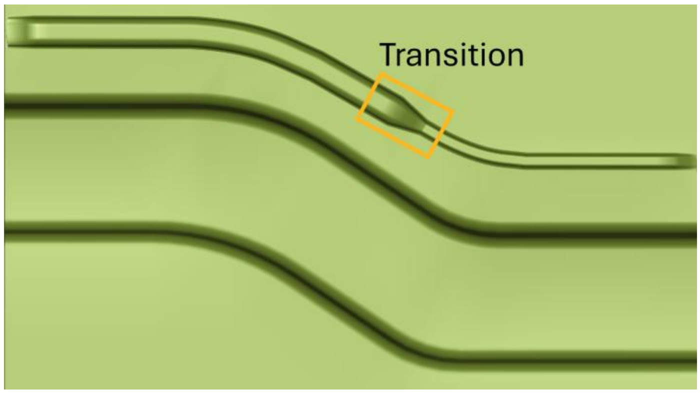
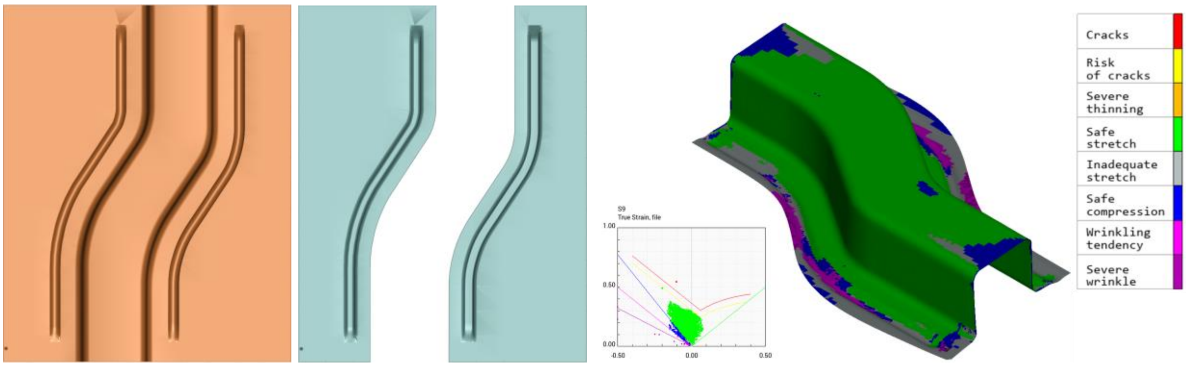


Fig.13: Illustration of "Transition Length" in 3D drawbead settings.

Fig. 14 shows the tool surfaces featuring 3D drawbead meshes generated from the 3D drawbead generator. The forming limit diagram from the simulation with the 3D drawbead mesh demonstrates that the material experiences good stretching during the forming process. This result suggests that the drawbead design effectively controls material flow and minimizes defects during the forming process, leading to optimal part quality and performance. The successful integration of the 3D drawbead mesh into the simulation process exemplifies the advancements in modeling capabilities within ANSYS FORMING.



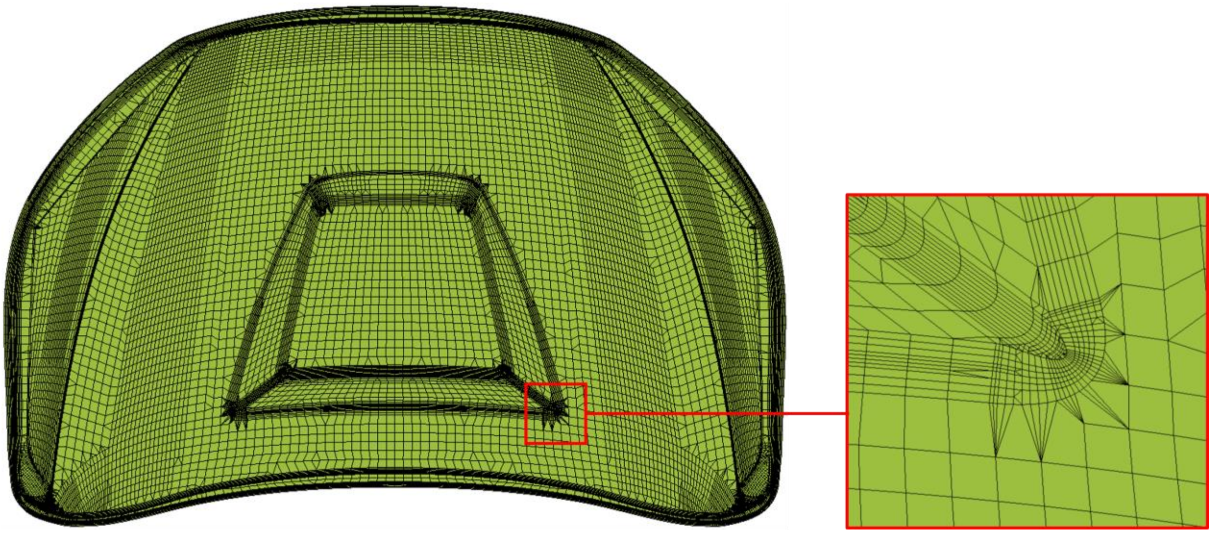
(a) Die with 3D bead (b) Binder with 3D bead (c) Forming limit diagram

Fig.14: Tool surfaces with 3D drawbead meshes and the corresponding forming limit diagram.

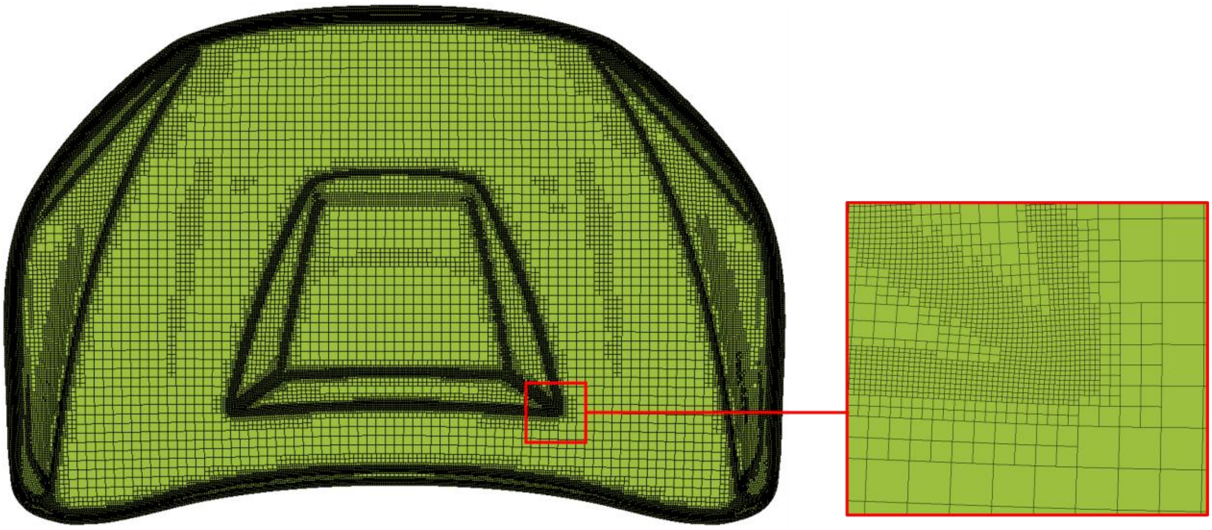
6 Adaptive Mesh Generation

In ANSYS FORMING, we have developed an innovative adaptive mesh regeneration technique for surface meshes, designed to enhance mesh quality while preserving the geometric accuracy. This method generates high-quality quadrilateral elements in the interior and allows for multi-resolution refinement along the boundary and surface, providing flexible control over mesh resolution in critical regions. Finer meshes can be applied to regions with greater geometric complexity, while coarser meshes are used in simpler regions, optimizing the balance between computational efficiency and simulation accuracy.

Fig. 15 compares an initial mesh of a part with the adaptively refined new mesh. The initial mesh contains excessively skewed elements in complex regions, which can compromise the simulation accuracy. In contrast, the adaptively refined new mesh consists of high-quality quadrilateral elements, leading to enhanced simulation performance.



(a) Initial mesh



(b) New mesh

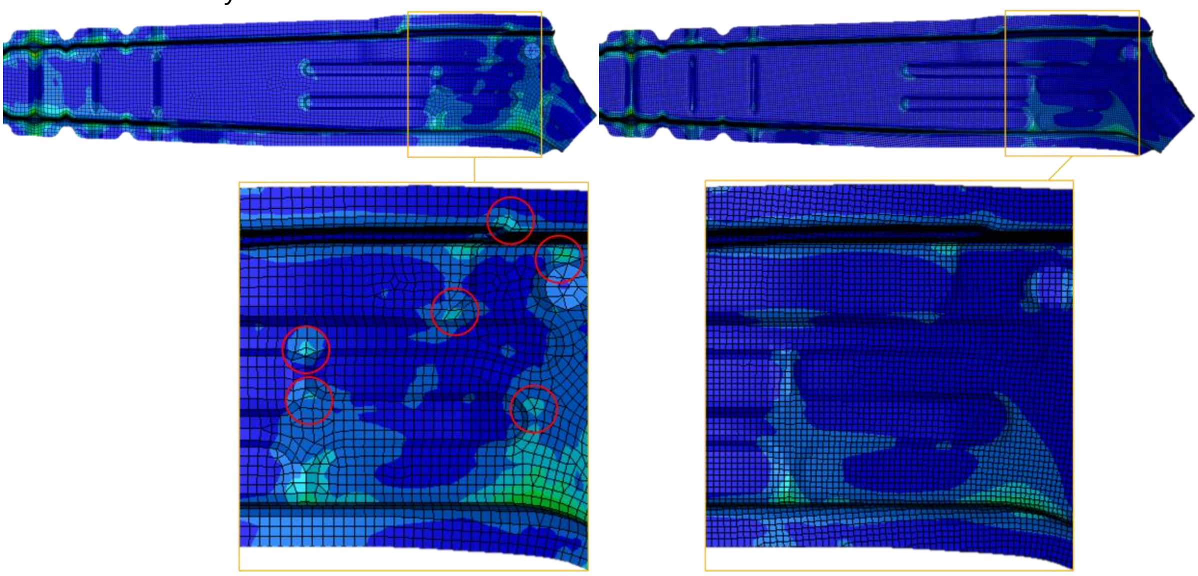
Fig.15: Comparison of the initial mesh and the adaptively refined new mesh.

7 Mesh Regeneration Applications

To demonstrate the effectiveness and potential of the adaptive mesh regeneration technique, we applied it to several applications, including one-step simulation, tool wear prediction, and hemming simulation.

7.1 One-Step Simulation

Fig. 16 compares the strain distributions from a one-step simulation using a mesh with a mix of quadrilateral and triangular elements in the interior and a new mesh with only quadrilateral elements in the interior. The result from the old mesh shows discontinuous strain distribution at the interface between quadrilateral and triangular elements, as highlighted in Fig. 16(a). In contrast, the solution field from the new mesh is smooth and continuous, providing a more accurate representation of the material behavior during the forming process. This comparison demonstrates the impact of element quality on the simulation accuracy.



(a) Old mesh (b) New mesh

Fig.16: Strain distributions from one-step simulation using: (a) the old mesh; (b) the new mesh.

7.2 Tool Wear Prediction

The second example focuses on tool wear prediction, which evaluates the durability of stamping tools. Fig. 17 shows the tool surfaces considered in this example, featuring a periodically arranged unit component. We will use a unit component to analyze the results.

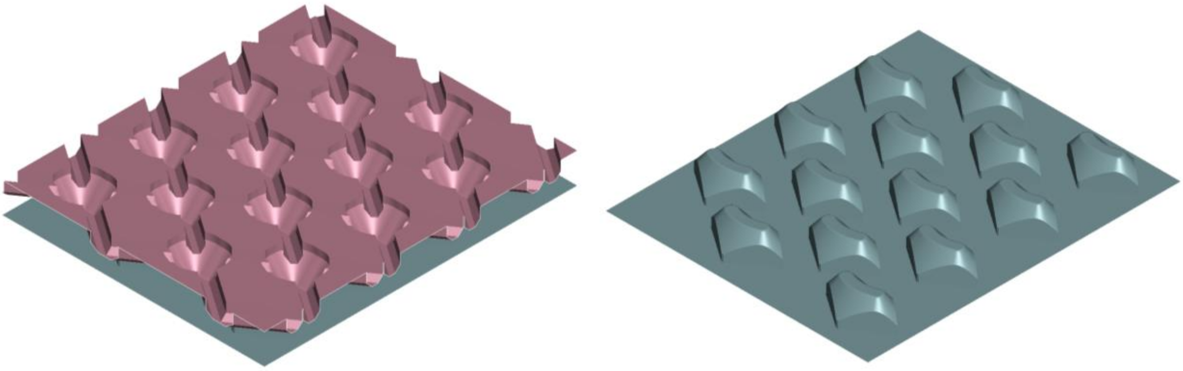
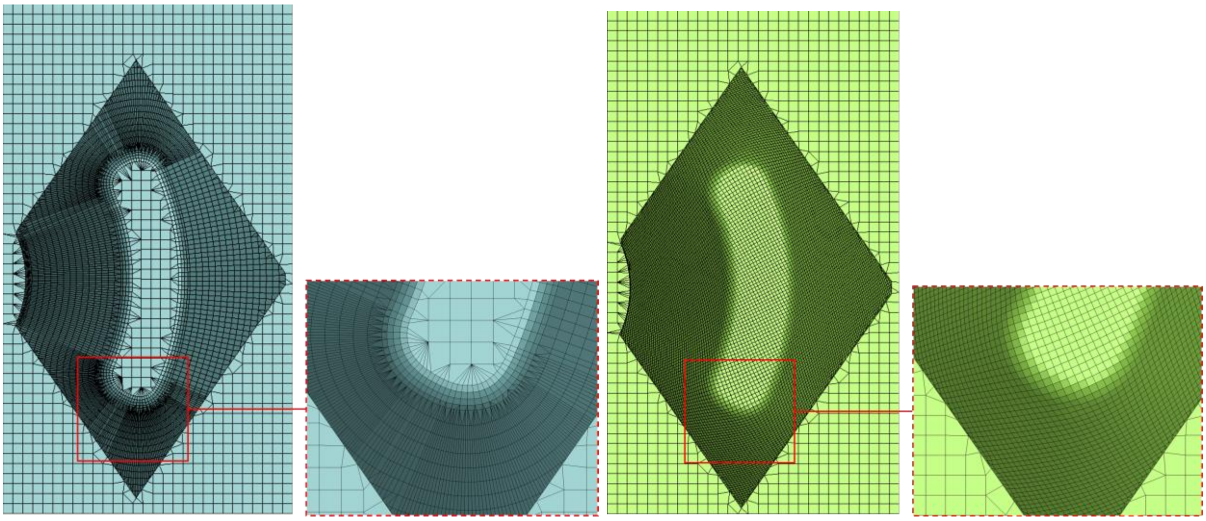


Fig.17: Tool surfaces considered in the tool wear simulation.

Fig. 18 compares the old mesh with the new mesh of the unit component. The old mesh contains excessively skewed elements, leading to an uneven distribution of friction energy density, while the new mesh consists of high-quality quadrilateral elements, ensuring a more uniform and accurate distribution of friction energy density.



(a) Old mesh (b) New mesh

Fig.18: Comparison between the old mesh and the new mesh of a unit component: (a) the old mesh; (b) the new mesh.

The cross-sectional profiles of the deformed tool after 20,000 stamps, obtained from the simulations using the old mesh and the new mesh, are compared against the experimental measurements in Fig. 19. It shows the deformation predicted from using the new mesh achieves a better agreement with the experimental measurements. This outcome again underscores the impact of element quality on the simulation accuracy. The improved mesh, characterized by high-quality quadrilateral elements, effectively captures the deformation process, leading to more reliable predictions.

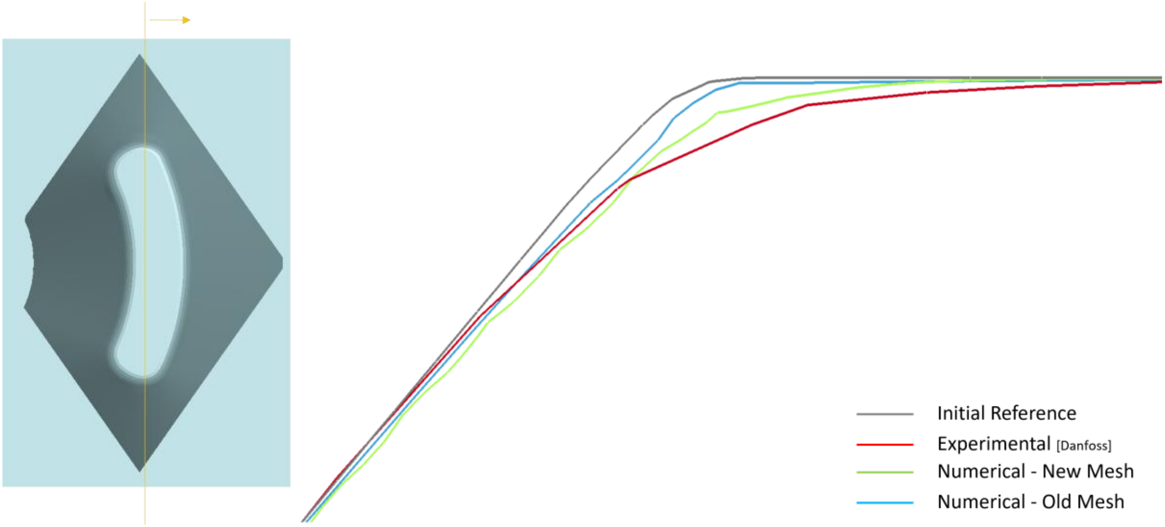
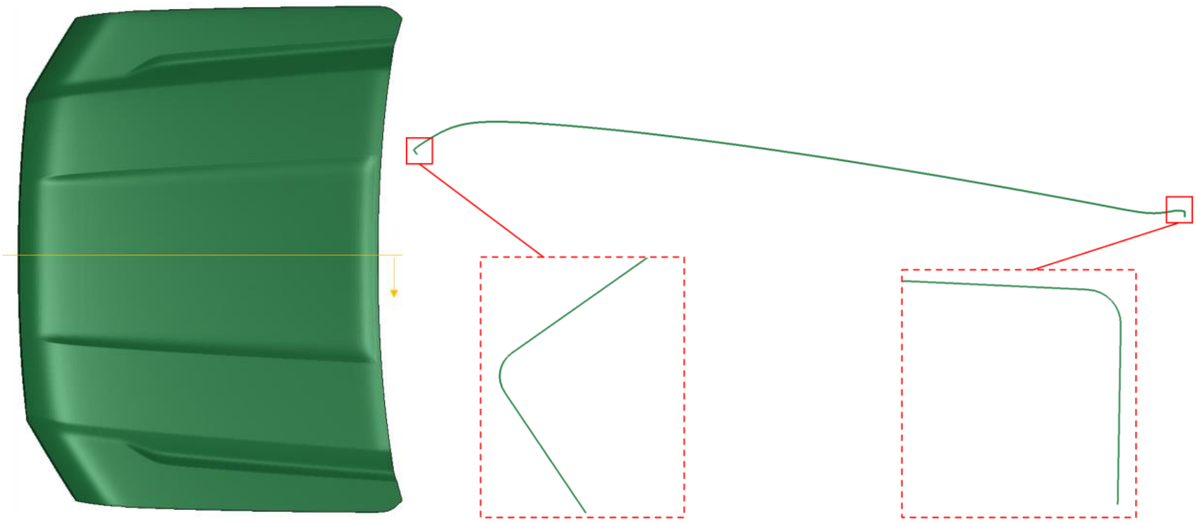


Fig.19: Comparison of the cross-sectional profile of the deformed tool from the old mesh, new mesh, and experiment.

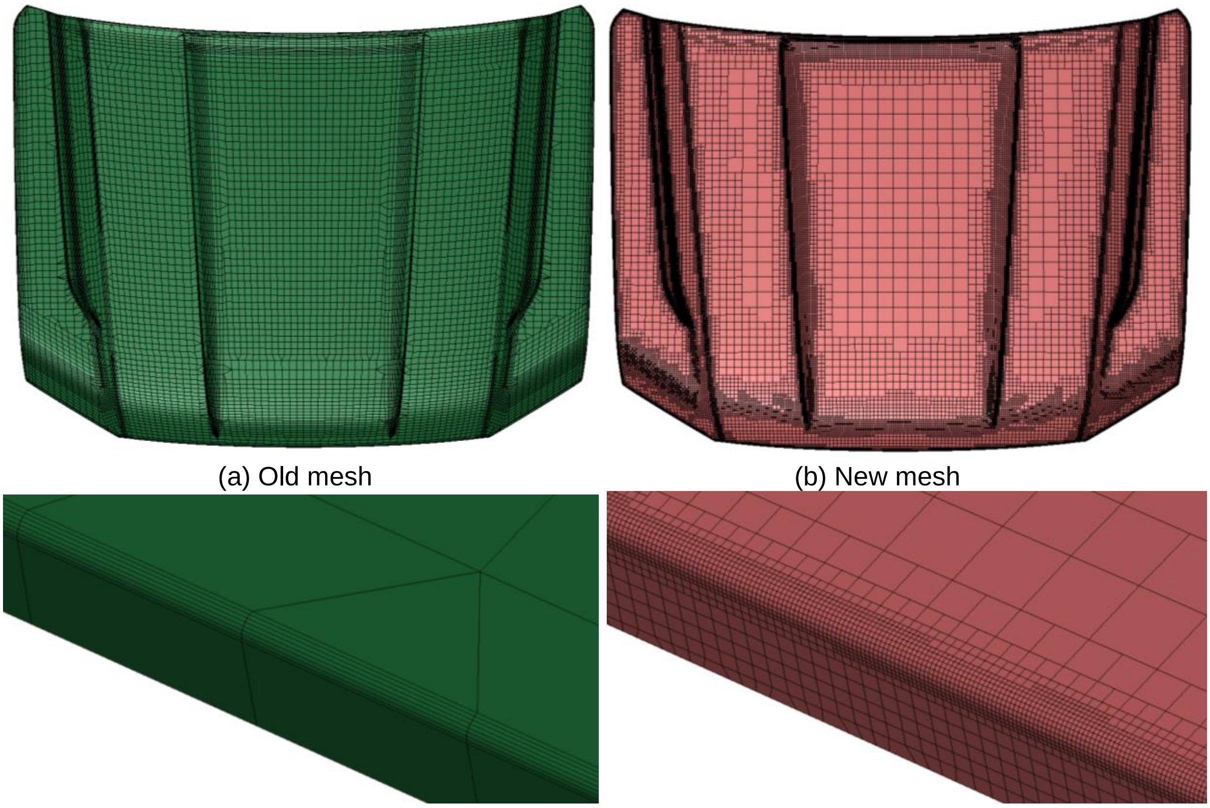
7.3 Hemming Simulation

In the last example, we demonstrate the potential of the adaptive mesh regeneration method in hemming simulations. Hemming is a forming process where the edge of a sheet metal is folded over to create a smooth and reinforced finish. Hemming simulations are crucial for predicting potential defects, such as distortion and springback, in assembled parts. Since hemming simulations often run in parallel with the die design process, tooling and forming details are usually unavailable and engineers commonly rely on the CAD model of the part to perform these simulations.
Fig. 20 shows the surface of the part under consideration, along with cross-sectional views of both the front and back ends, highlighting high-curvature regions near edges. Achieving accurate hemming predictions requires a high mesh resolution around the edge region. However, it is challenging for structured meshes to capture detailed geometries around high-curvature regions and global refinement can result in a large number of elements, significantly increasing the computational cost. In contract, our adaptive refinement approach allows for finer mesh resolution in the high-curvature regions near edges and coarser mesh resolution in simpler areas, as shown in Fig. 21, leading to more accurate and efficient hemming simulation. This example highlights the versatility of adaptive mesh regeneration in addressing the challenges of complex forming processes.



(a) Part surface (b) Cross-sectional view

Fig.20: Part surface considered in the hemming simulation and its cross-sectional view.



(a) Old mesh (b) New mesh

(c) Front edge region of the old mesh (d) Front edge region of the new mesh

Fig.21: Comparison between the old mesh and the new mesh: (a) the old mesh; (b) the new mesh; (c) the front edge region of the old mesh; (d) the front edge region of the new mesh.

8 Summary

We have developed innovative features to enhance the 3D drawbead design and meshing capabilities of ANSYS FORMING for more accurate and efficient forming simulations. These features include a 2D drawbead profile generator, drawbead simulator, and 3D drawbead generator, which streamline the drawbead design process by allowing users to create 2D drawbead profiles, estimate drawbead forces, and generate 3D drawbead meshes. This eliminates the need for repetitive geometry adjustments and accelerates the drawbead design cycle.

Additionally, the novel adaptive mesh regeneration feature enhances mesh quality by generating high-quality quadrilateral elements and applying adaptive refinement in complex areas. This improves simulation accuracy while minimizing computational cost. The effectiveness of this method is demonstrated through improved prediction accuracy in one-step simulations, tool wear analysis, and hemming simulations based on CAD models, highlighting its wide-ranging potential.