Integrated Workflow for Material Selection and Vibration Analysis of Automotive Parts Including Forming Effects

Dharanivendhan T[1], Kang Shen[2], Mehul Muradia[3], Akhil KS[4], Ansys part of Synopsys

1 Introduction

Forming operations involve a complex interplay of parameters that must be optimized to achieve a defect-free product while satisfying design requirements. The process typically begins with material selection, which is often performed during the initial part of product design stage. Following material selection, aspects like draw bead design, tool geometry, and process conditions are iteratively refined through simulation and validation to ensure the performance and manufacturability of the formed part.

In this study, we have tried to address the challenges in material selection, forming process optimization and considering the forming effects in the downstream analysis (Modal) with the help of a car hood model shown in Fig.1:. The part also needs to be sufficiently stretched to successfully perform spring back compensation as explained in [2] without crack and the frequencies need to be in a specific range. The workflow could be extended to perform harmonic response, denting or fatigue analysis which are sensitive to thickness variation and prestress effects in the parts.

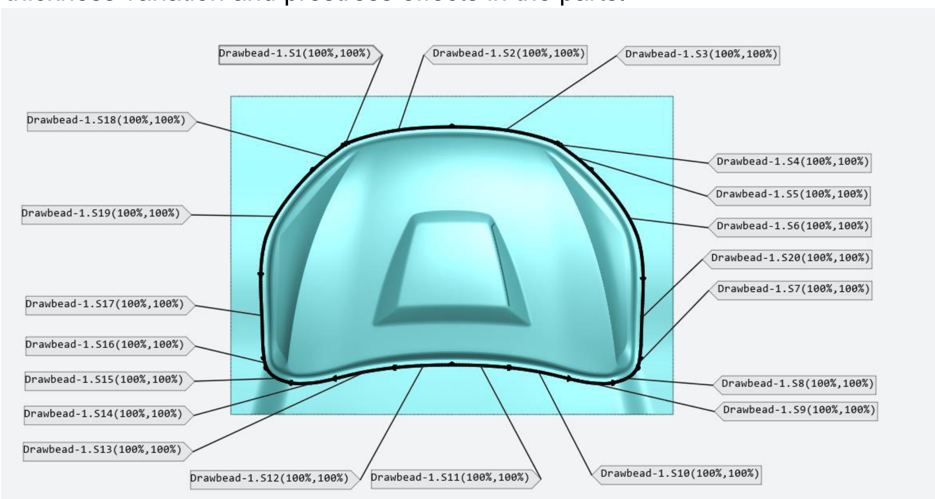


Fig.1: Various Drawbead segments along the periphery of the Hood

2 Methodology

The sequence of analysis performed in shown in Fig2. In all the 3 stages optiSLang has been integrated for optimization and automating the workflow. Further, the procedure to transfer the results from LS-DYNA to mechanical to perform other advanced analysis has been explain in 3.4.2.

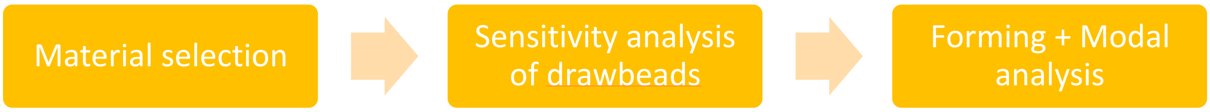


Fig.2: Sequence of analysis performed

2.1 Approach for material selection

2.1.1 Material Selection and Optimization Strategy

There are well defined guidelines to choose materials based on formability parameters as later explained in the section. It is essential to have an idea of what kind of stress- stain states the formed part might experience. For example, in the case of deep drawing, it is preferable to choose material with higher R value (draw quality materials). For a generic case where it not straight forward to determine the stress-state, One-step forming simulation provides valuable insights such as, stress–strain fields, thickness distribution, formability in forming-limit diagram (FLD) plots and the initial blank shape very fast and reasonably accurate. These outputs enable rapid material screening before detailed die design. Additionally, for automotive outer panels, stamping operations typically employ draw beads to control material flow and stretch(insufficient stretch can also results in more springback). The perfect drawbead forces(no crack) is dependent on the specific material properties. For examples as illustrated in Fig.3:.

MAT1 (($\sigma_y \approx 200\ MPa$): At 75% Restraining force, insufficient stretch.

MAT2 ($\sigma_y \approx 400\ MPa$): At 75% Restraining force sufficient stretch is achieved.
⇒ Hence, restraining force should be included as an optimization parameter alongside material ID.

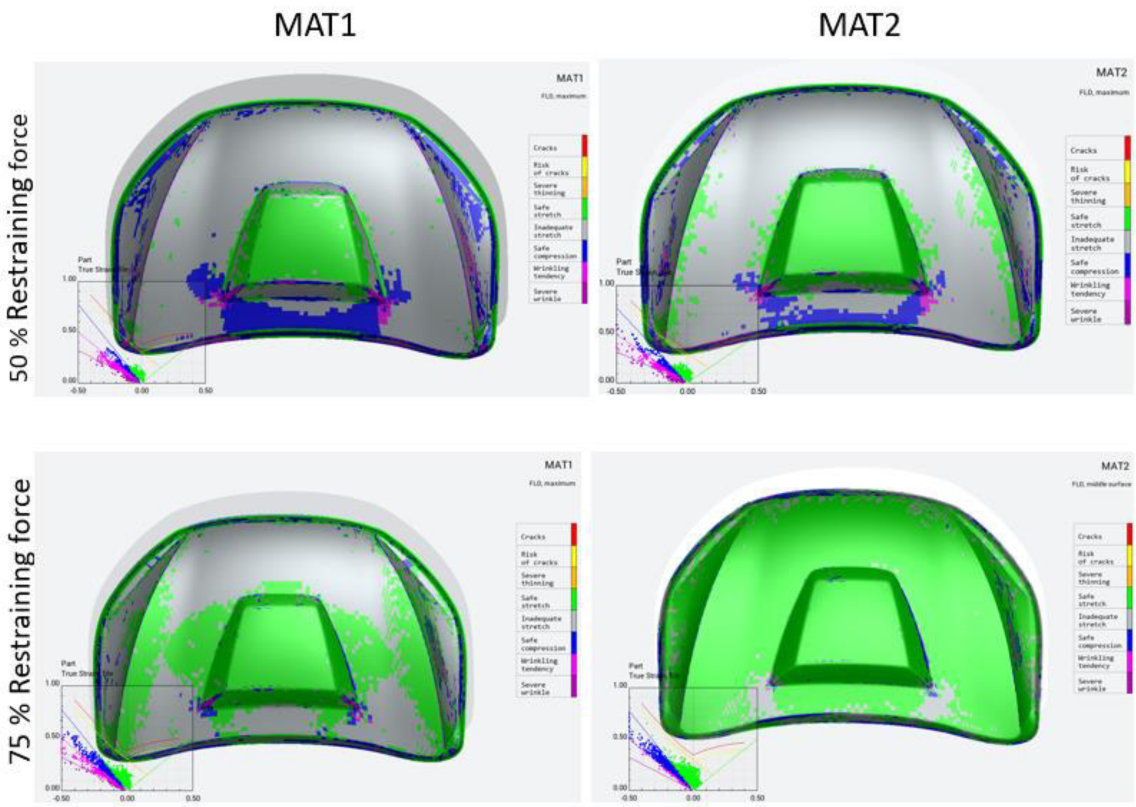


Fig.3: Comparison of FLD plots of MAT1 and MAT2

In Ansys Forming, the restraining force is computed internally from tensile strength and sheet thickness [5] and can be defined in GUI as shown in Fig.4:.

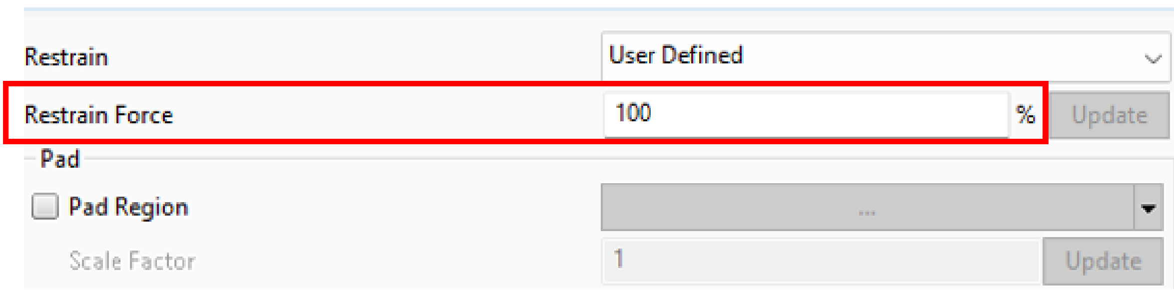


Fig.4: Restrain force option in Ansys Forming

2.1.1.1 Strategies for Material Selection

Material Selection Strategies and draw bead Optimization for Sheet-Metal Forming

1) Strategies Evaluated
S1 — Parameterize material properties (per [4]).
Optimize directly on material parameters like yield strength (σ_y), ultimate strength(σ_t), Anisotropy parameters R_{00}, R_{45}, and R_{90}, hardening curve parameters $E_0, n\ and\ K$ as (Swift law parameters as shown in below equation).

$$\sigma_Y = K(\varepsilon_0 + \varepsilon_p)^n$$

S2 — Discrete material sweep.
Treat Material ID as a design variable in optiSLang and optimize over a curated library (e.g., 20–30 grades). Use a Python script to rank short-listed grades by additional formability metrics once a promising range is found.

2) Challenges with Full Parameterization (S1)

- Physical realism constraints. The optimizer can propose non-physical combinations (e.g. $\sigma_y = 100\ MPa$ and $\sigma_t = 900\ MPa$). Additional side-constraints (e.g., say $\sigma_t = 1.5\sigma_y$) are required to keep candidates realistic.

- FLD parameterization. Empirical FLD relations (e.g., ThyssenKrupp for certain steels, as implemented in Ansys Forming; see Fig.5: and [3]) require extra fitting parameters and are not universally applicable across all steel types. This increases complexity and uncertainty in prediction of formability.

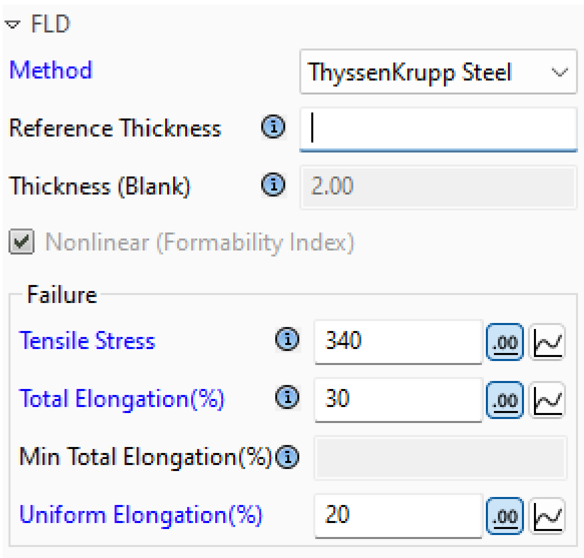


Fig.5: FLD of ThyssenKrupp Steel in Ansys Forming

3) Advantages of Discrete Sweep (S2)

Optimizing over real datasheet-based materials avoids the need of adding constraints as explained previously and yields directly procurable grades. Although iterating over 20–30 materials can be time-consuming, optiSLang handles this efficiently by treating Material ID as an integer variable.

4) Hybrid Workflow Adopted

- One-Step screening: Run a one-step simulation with four representative yield-strength classes (200, 400, 600, 800 MPa) to coarsely localize the viable strength range for the target geometry. Yield strength is chosen as a criterion for grouping materials as the design requirements ften demand highest possible strength of material yet ensuring formability of the part. Here yield strength is chosen as a parameter as it correlates reasonably well with formability; higher yield strength usually means lower formability. The assumption does not take into consideration the processing history- bake hardening, aging, hot or cold rolled. However, the current approach can be further extended by choosing anchor material based on processing history and providing more useful and realistic material suggestions.

- Focused library search: Within the identified range, use a Python script to query the material database and rank candidates by secondary formability indicators.

- Joint optimization: In optiSLang, co-optimize Material ID and draw bead restraining force to achieve target stretch without cracking and with acceptable strength.

5) Ranking Criteria (from standard texts like [6])

The following set of parameters are secondary as they determine the behavior of material on the onset of plastic deformation. Therefore, once a group of materials with similar yield strength is identified, these parameter could be used to further sort the materials to rank their formability.

Hardening exponent(n): Higher delays localized necking and improves formability—preferable for complex geometries.

Strength coefficient: Higher implies a stronger final part but increases forming load/tooling force.

Normal anisotropy (Lankford - $\bar{r}$ value): Higher increases resistance to thinning; prioritize when thickness control is critical.

2.1.1.2 Material Selection

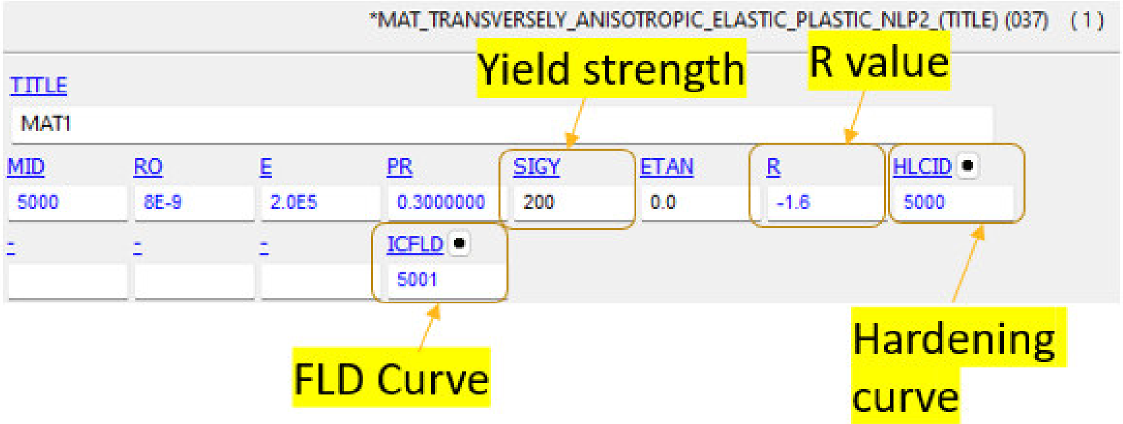
Once the optimization identifies the best candidate among the four discrete materials, a **Python-based selection script** queries a material database to identify commercially available materials with similar yield strength.

In this work, a simple version of the script has been developed which can fetch data from **`*MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC_NLP2_TITLE(Fig.6:a)`** and extract the forming related parameters as explained in Fig.6:b. The hardening curve and the FLD are converted into scalar parameters – The hardening curve are fitted using SciPy python libraries to get strain-hardening exponent (*n*), strength coefficient (*K*) and hardening parameter (ε_0) . ε_0 is not reported as it is not a decisive formability parameter, The FLD curve is converted into a single value by averaging the Ordinate which is measure of how far the centroid lies, higher the centroid higher the formability limit is assumed. The extracted properties (Rounded off for confidentiality) of anchor materials are shown in Table 1:.

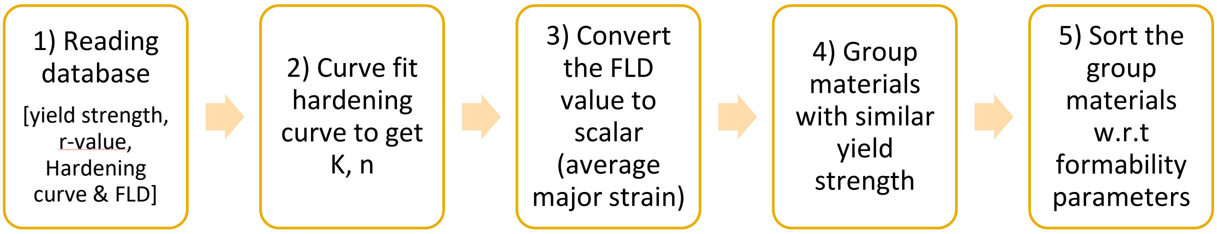
Table 1: Properties of anchor materials(Rounded -off)

	$\bar{r}$	Yield strength(MPa)	K	N	FLD
1	1.593	200	550	0.19	0.53
2	0.864	400	1000	0.17	0.48
3	0.806	600	1400	0.16	0.47
4	0.85	800	1600	0.09	0.37

Once the optimized value of yield strength is provided by optiSLang, the value can be entered in python code and the code shows materials with similar properties along with other formability parameters as shown in Fig.6: c.



a - **`*MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC_NLP2_TITLE`**



b – Broder steps in python script

Input of python script

```
output = Sorting_material(df, 250,"R")
```

Input from optiSLang

Output of python script

	R	Yield_value	K	N	FLD
11	1.105	297.0	626.5576	0.1836	0.3973
3	1.180	266.0	569.2787	0.1827	0.4002
4	1.240	273.0	549.4499	0.2082	0.4262

c – Broder steps in python script

Fig.6: Python script logic for material selection

2.1.1.3 Demo on material selection

Two different sample models are chosen to demonstrate the material selection logic.

- S-Rail

- Car Hood model

The S-Rail model was used as a validation case in the material selection workflow, given its suitability for deep drawing where materials with high formability and low yield strength are preferred. The optimization was framed as a Mult objective problem in optiSLang, aiming to maximize minimum thickness and minimize crack occurrence, helping identify materials that balance thickness and formability effectively.

$$Objective: Maxmize(Min\ thickness)$$
$$Minimize(Crack)$$

For the case of Hood simulation, the objective function and the constraints applied to the model are shown in Fig.7:7. The formulation of the objective function and constraints are shown in below equation. In the case of Hood, the yield strength of the material is maximized subjected to formability constraints.

$$Objective = Maximize(\sigma_y)$$
$$\%\ Crack \leq 0$$
$$\%\ Risk\_of\_crack \leq 0.2$$
$$\%Safe\_stretch \geq 75$$

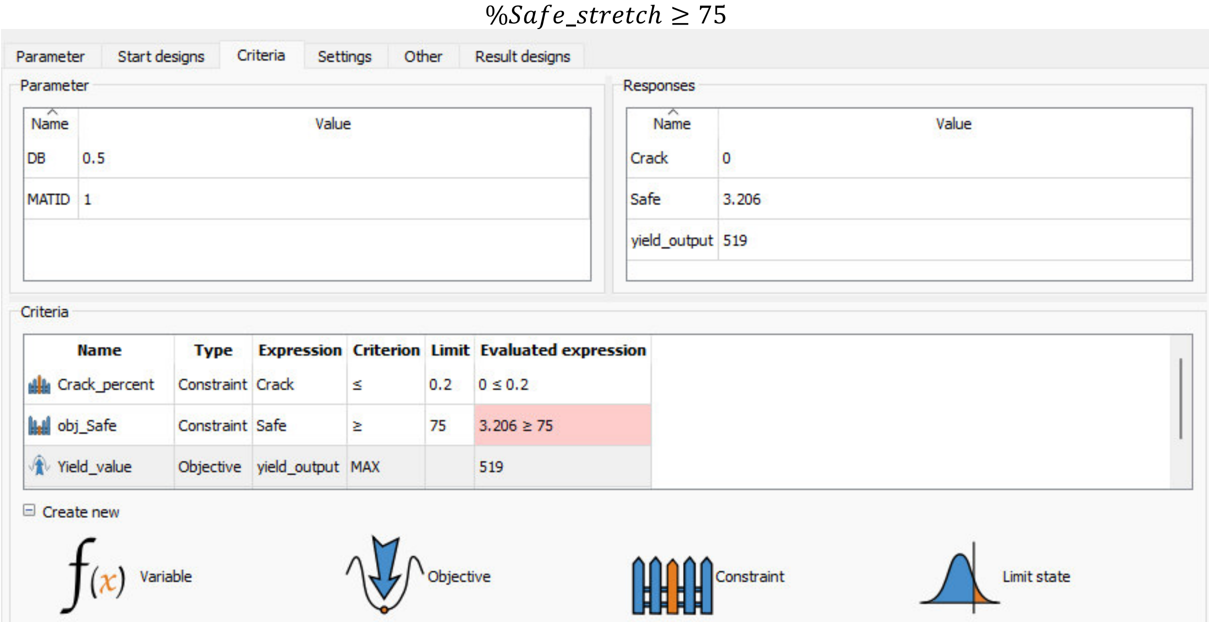


Fig.7: Objective and Constraints of optimization of Hood model

2.2 Forming optimization for Downstream analysis

After the material selection from previous section, sensitivity analysis on the draw beads are performed to identify the critical draw beads and then a full optimization is performed to include the forming effects.

Sensitivity analysis:

After the material selection process, the optimised material is fixed, further a sensitivity analysis is performed to identify critical draw beads that may affect the stretch and cracking of the component. The restraining force % of all the 20 draw beads as shown in Fig.8:8 is parameterised, and a sensitivity analysis was performed to identify critical draw beads.

```
*PARAMETER_EXPRESSION
RDB1      0.5
RDB2      0.65
RDB3      0.7
RDB4      0.4
RDB5      0.5
RDB6      0.65
RDB7      0.7
RDB8      0.4
RDB9      0.5
RDB10     0.65
RDB11     0.7
RDB12     0.4
RDB13     0.5
RDB14     0.65
RDB15     0.7
RDB16     0.4
RDB17     0.5
RDB18     0.65
RDB19     0.7
RDB20     0.7
```

Fig.8: Drawbeads as parameters.

Forming plus Modal analysis:
The Drawing and Trimming operations are setup in Ansys Forming and the Springback plus modal analysis are performed using LS-DYNA keywords (shown in Fig.9:) and are explained in detail in 3.4.1. It is recommended to perform a springback analysis before modal analysis as it is found to provide the modal frequencies more robustly.

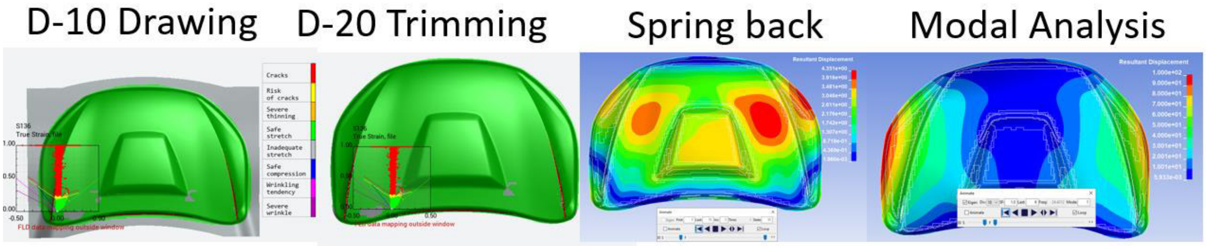


Fig.9: Sequence of analysis in Forming plus modal analysis

The output parameters are read from FLD Report, eigout and thickness report(saved from History plots) as shown in Fig.10:.

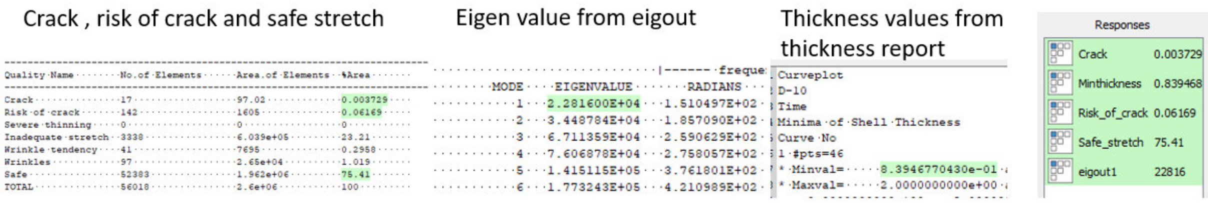


Fig.10: Output responses for the optimization analysis

In the final forming plus modal analysis, only the critical drawbeads and additionally the sheet thickness (signifying geometry change) is treated as design parameter, while the non-critical drawbeads are assigned as constant values. These constant values are derived from the results of the sensitivity analysis. Specifically, the tabular data from the sensitivity iterations is sorted in increasing order of crack severity as shown in Fig.11:1. From this sorted data, the iteration corresponding to zero crack is identified. The drawbead values from this zero-crack iteration are then extracted and used as fixed constants for the non-critical drawbeads in the final analysis. This approach ensures that the optimization focuses only on the most influential parameters(critical drawbeads)and thickness—while maintaining the stability provided by the optimal non-critical parameters.

	DB_14	DB_15	DB_16	DB_17	DB_18	DB_19	DB_2	DB_20	DB_3	DB_4	DB_5	DB_6	DB_7	DB_8	DB_9	Crack	fld_report
1	0.43	0.62	0.92	0.24	0.41	0.99	0.43	0.24	0.99	0.67	0.45	0.89	0.41	0.5	0.79	0	70.07
2	0.31	0.21	0.45	0.65	0.43	0.36	0.7	0.94	0.87	0.82	0.48	0.75	0.28	0.24	0.77	0	68.92
3	0.71	0.54	0.54	0.49	0.31	0.92	0.75	0.21	0.49	0.94	0.89	0.8	0.4	0.59	0.85	0.005696	74.68
4	0.43	0.43	0.87	0.84	0.21	0.67	0.41	0.24	0.96	0.48	0.58	0.5	0.5	0.21	0.53	0.005717	71.39
5	0.94	0.84	0.28	0.99	0.28	0.75	0.96	0.92	0.26	0.55	0.5	0.41	0.31	0.99	0.82	0.02037	69.53

Fig.11: Results of Sensitivity analysis of drawbead optimization

Parameterizing thickness:
Apart from making the Thickness as a parameter in **`*SECTION_SHELL,`** the following changes needs to be done in ANSYS Forming deck. Please check Fig.12:2

- the contact offsets in the forming and draw bead contacts needs to be parametrised with blank thickness
- The tool motions have parameters for ramping up and ramping down the tool velocities. The ramp up and ramp down distances needs to be also parametrised equal to blank distance.
- The Thickness values in Auto positioning cards needs to be parametrised and set equal to 1.1 times blank thickness to maintain sufficient gap.

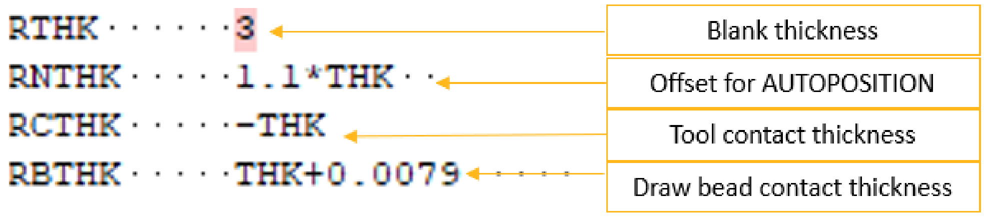


Fig.12: Parameter Expression for thickness, thickness offsets and tool positioning

Objective function and constraints:
The following objective function and constraints were used for optimization.

$$Objective = Maximize(Safe\ stretch)$$
$$Objective = Maximize(First\ frequency)$$
$$\%\ Crack \leq 0$$
$$\%\ Risk\_of\_crack \leq 0$$
$$\%Min\_thickness \geq 1.8$$

2.3 OptiSLang

2.3.1 Solver Wizard setup

The solver wizard in optiSLang is first setup to automate material selection workflow as shown in Fig.14:4.
The solver module contains of three nodes – DYNA Input, Solver and ETK module.
DYNA Input node:
The input files generated by Ansys Forming are read using Dyna Input node and the input parameters are located as shown in Fig.13:3. The input file from Ansys Forming needs to be modified by including **`*PARAMETER_EXPRESSION`**. Correspondingly the variable needs to be referred in appropriate sections in the input file using "&" in **`*PART`** and **`*CONTROL_FORMING_ONESTEP`** as shown in Fig.13:.

```
*PARAMETER_EXPRESSION
$#    prmr
IMATID    1
RDF       .25
```

```
*CONTROL_FORMING_ONESTEP
$#   option      unused      autobd
          6                     &DF
```

```
*PART
S2
$#       pid     secid       mid     eosid      hgid      grav    adpopt      tmid
           2         1    &MATID         0         1         0         0         0
```

Fig.13: Parameters for material selection

Solver Module:

Step 1: Python Script – FLD Curve Update

- Purpose: Dynamically modify the FLD curve number in the LS-PrePost macro (cfile) based on the selected Material ID.
- Functionality:
  - Reads the Material ID from the input deck.
  - Updates the macro to use the corresponding FLD curve for that material.
  - Ensures correct postprocessing alignment with material-specific forming limits.

Step 2: LS-DYNA Simulation Execution

- Method: Run via CMD commands.
- Input: Modified input deck from Ansys Forming.
- Output: Simulation results d3plot files.

Step 3: LS-PrePost Macro Execution

- Tool: LS-PrePost using recorded cfile macros.
- Purpose: Generate FLD Report, thickness report and extract forming-related metrics.
- Automation: Macro rerun is triggered post-simulation to ensure consistent postprocessing.

Step 4: Python Script – Yield Strength Extraction

- Purpose: Read yield strength (σ_y) from the material input file.
- Output: Writes the value to a text file.
- Integration: optiSLang ETK module reads this file as an output variable for optimization.

ETK Module:

- The ETK module reads import parameters like %Crack and %Safe stretch from FLD report, Min thickness from thickness report (applicable for S-Rail), also reads the yield stress from text file generated by python script as explained previously.

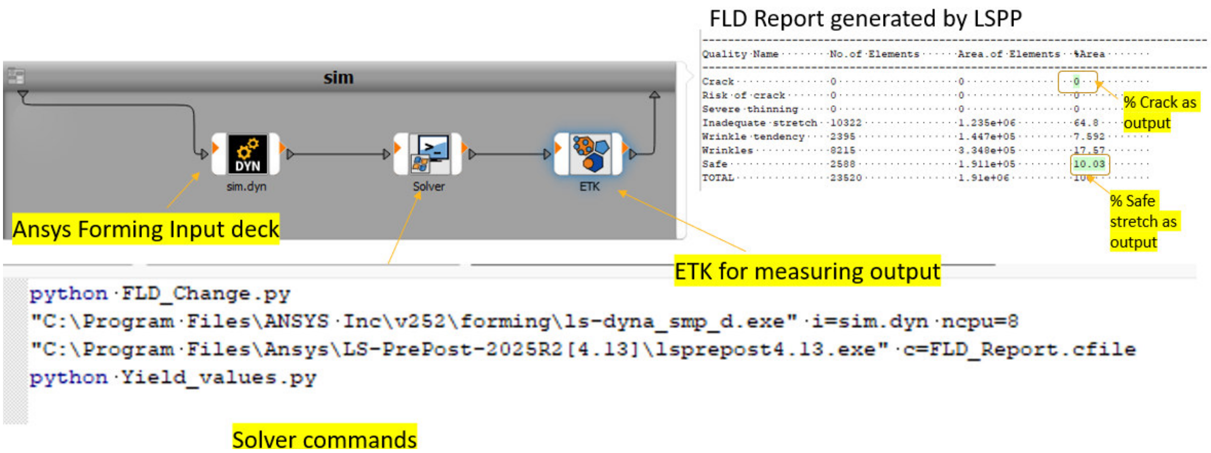


Fig.14: optiSLang Solver wizard for material selection

2.3.2 Overview of optiSLang

A brief overview of the methodology used for optimization is explained in this section.

2.3.2.1 Sensitivity Analysis

Sensitivity analysis (SA) is a key technique in computational modeling used to understand how changes in input variables affect model outputs. It helps identify which variables have the greatest impact on optimization goals, allowing designers to focus on the most influential parameters and reduce the number of design variables. Variance-based sensitivity analysis, commonly used as a pre-processing step, quantifies how much each variable contributes to changes in model responses within their defined ranges.

2.3.2.2 Methodologies for Quantifying Variable Importance

The general flow of sensitivity analysis involves defining variables and responses, scanning the design space using Design of Experiments (DoE) or sampling methods, evaluating samples via a solver, generating approximation models (Reduced Order Models/ metamodels), and subsequently estimating variance-based sensitivity indices.

• **Metamodel of Optimal Prognosis (MOP) Sensitivity:** This approach determines the optimal input variable subspace and the most suitable approximation model (e.g., polynomial or Moving Least Squares) using the **Coefficient of Prognosis (CoP)**.

For the current case, we have used Advanced Latin Hypercube(ALH) as the Design of Experiment schema due to its efficiency in managing high-dimensional uncertainty spaces and its robustness against sampling bias. Also, ALH is suitable for models with <50 parameters, which makes it a good choice for the current use case.

Additionally, we had used the adaptive metamodel of optimal prognosis (AMOP). Using this, optiSLang was able to start with very few DoE sampling points (34), and after this initial DoE, AMOP automatically creates MOP for the outputs. It identifies regions where the metamodeling is favorable and regions where new observations could improve quality. Based on this insight, AMOP automatically runs new simulations for subsequent iterations. In this way, AMOP redefines the DoE where needed to achieve the best metamodel quality, requiring less manual input and fewer simulations.

Finally, sensitivities of each metamodel are obtained and the metamodel for each output variable is created using one of the following methods- Linear Regression of order 2 (no mixed terms, BoxCox)/Isotropic Kriging

Multi-Objective Optimization

Most real-world optimization problems inherently include more than one conflicting objective. The general formulation requires minimizing a vector of objective functions subject to constraints and variable bounds.
• **Interactive Methods (Pareto Optimization):** When preferences are not defined *a priori*, interactive methods such as Pareto optimization are employed. Pareto optimality utilizes the concept of **Pareto dominance** to create a partial order among design vectors. A solution is Pareto optimal if no other solution can improve one objective without causing a deterioration in at least one other objective. The goal is to find the **Pareto set** (non-dominated subset of solutions) and the corresponding **Pareto frontier** in the objective space.

Optimization Strategies and Algorithms

The efficiency of optimization algorithms often diminishes rapidly with an increasing number of design variables. Thus, modern methodologies frequently integrate Sensitivity Analysis (SA) to dramatically reduce the design space prior to optimization, identifying variables that contribute most to the desired goal improvement. The primary types of optimization algorithms available include gradient-based, response surface-based, and population-based methods.

These methods replace time-consuming solver calls by utilizing mathematical surrogate functions, or metamodels, to approximate model responses.
• **Metamodel of Optimal Prognosis (MOP) Optimization:** Using the MOP(Response surface base approach) approximation for optimization is highly recommended as a first step due to its reliable estimation of approximation quality via the **Coefficient of Prognosis (CoP)**, which is superior to the Coefficient of Determination (CoD). The MOP smooths solver noise, offering more stability than gradient-based methods, and facilitates the use of global optimizers (e.g.,

evolutionary algorithms) on the resulting approximation function to solve non-convex problems. Designs identified via MOP-based optimization often surpass those found during preceding sensitivity analyses.

3 Results

The results of material selection, sensitivity analysis of Draw beads and forming plus modal analysis are covered in upcoming section.

3.1 Material selection

3.1.1 S-Rail

The S-Rail is formulated as a multi-objective function as explained previously, For multi-objective functions optiSLang creates a pareto front(check 2.3.2) which represents the optimal tradeoff between the two objectives as shown in Fig.15:5. The x-axis represents the thickness values, and the y axis represents the % of crack. One of the optimum designs obtained is MAT1 and the corresponding % Restrain forces is 0.65. The thickness values and safe stretch values are reported to be around 0.78 mm and 65% Safe stretch as shown in the right image of Fig.15:5. The values reported here are the values calculated by the Reduced Order Model.

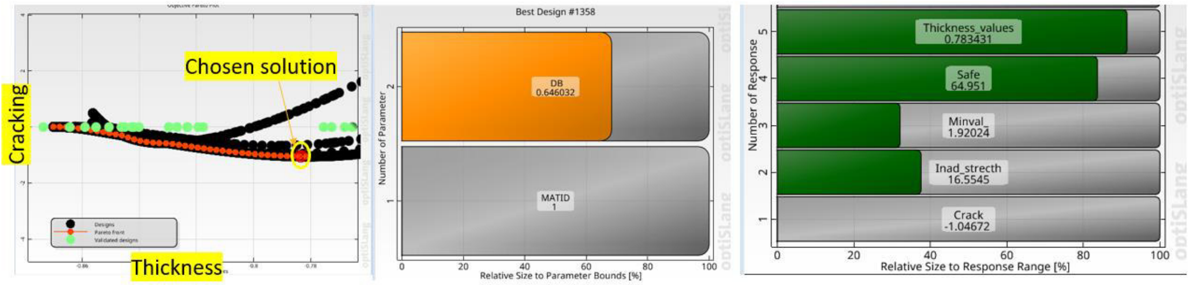


Fig.15: Pareto front diagram in Mult objective function; Best design; Response surface values

For validation of the optimum design values obtained through optiSLang, the calculation is re performed in Ansys Forming and the results are shown in Fig.16:6. The minimum thickness is observed to be around 0.82 mm and the component appears to be mostly in safe stretching condition.

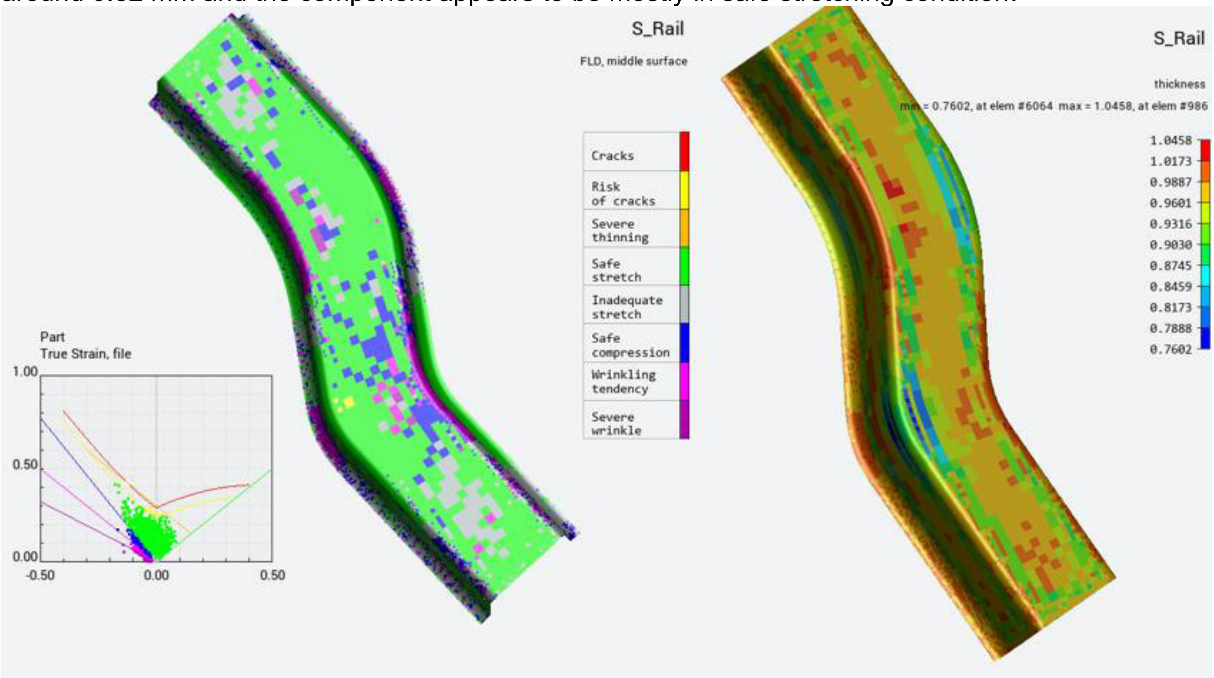


Fig.16: Thickness plots and FLD plots of S-Rail with MAT1

3.1.2 Hood Geometry

Opti Slang gives the solution of the best material as MATID – 2(check Fig.17:) which has a yield strength of around 400 MPa as shown in Table 1:. The FLD plot of the optimized results are shown in Fig.18:8. The material is in safe zone for most areas with very little cracks which can be improved in detailed forming design. The minimum thickness of the material is found to be around 1.5 mm as shown in Fig.19:

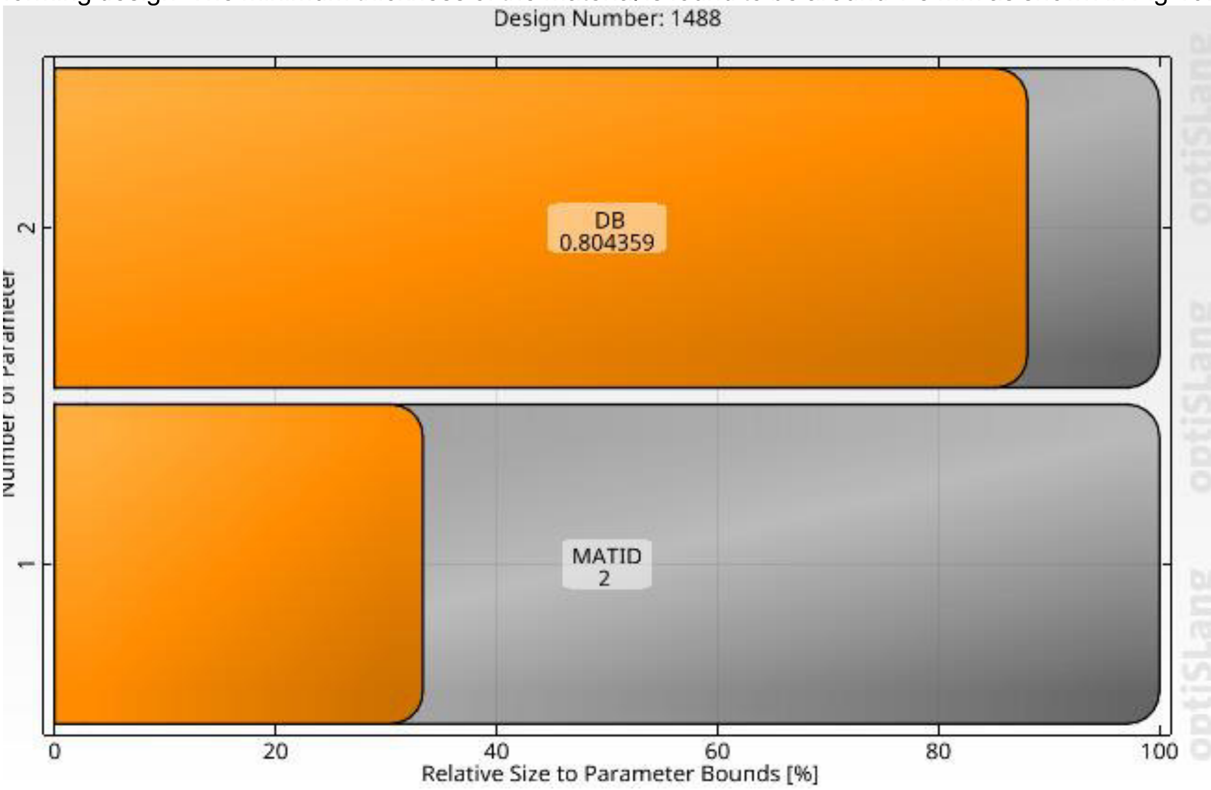


Fig.17: Optimized design output from Material selection criteria

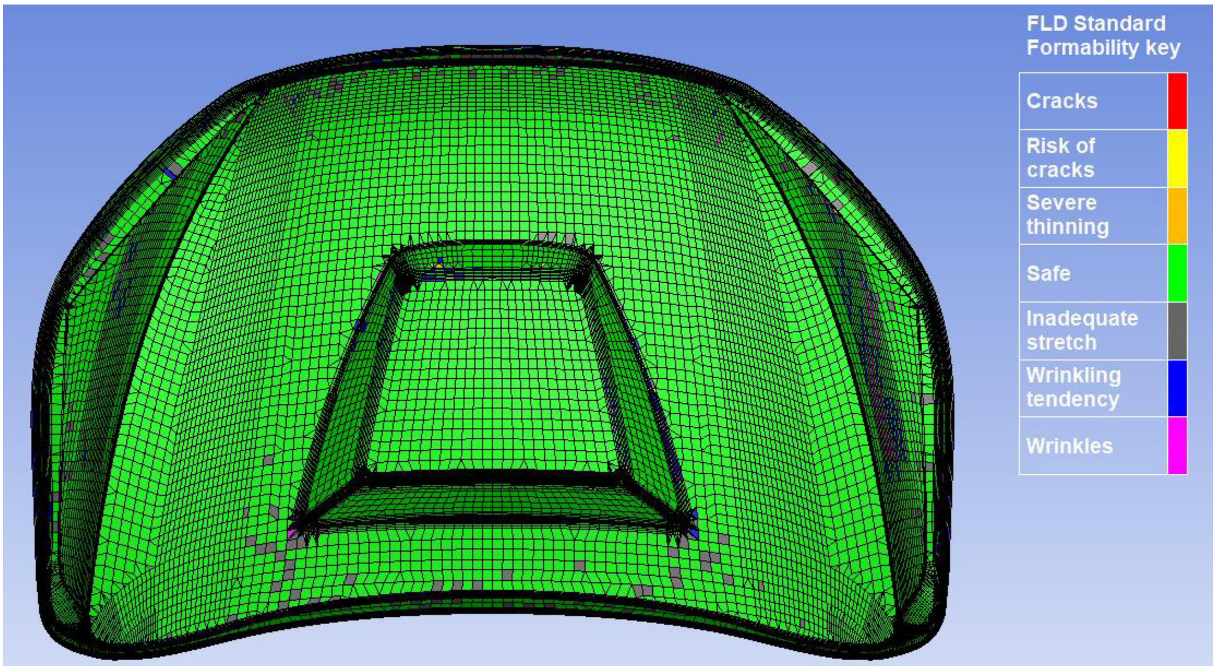


Fig.18: FLD plot of optimized design with MATID 2

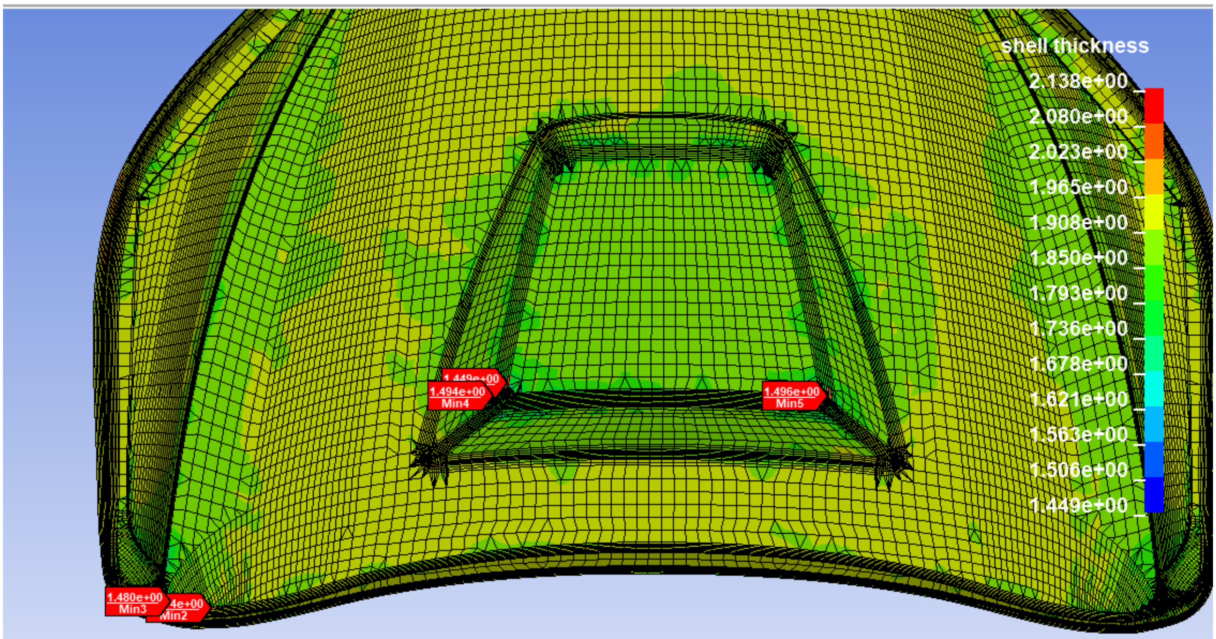


Fig.19: Thinning observed in sharp corners in hood model

The python script is now used to choose material having yield strength closer to $\sigma_y = 400\ MPa$. MATID 2 has R value of around 0.86 and "Average FLD" of around .48(Refer Table 1:). Mat 9 seems to have properties closer to the specifications w.r.t. to "R" and FLD. Mat 9 is used for further detailed forming analysis in the next section.

	R	Yield_value	K	N	FLD
9	0.7923	428.163	1053.5493	0.1760	0.4997
14	0.8475	463.500	1337.4432	0.2205	0.2849
12	0.9650	410.000	745.9194	0.1738	0.3810
8	1.0250	460.000	644.6578	0.1216	0.3544
10	1.0430	486.000	1248.5718	0.1287	0.3391

Fig.20: Material suggestion from python script for hood model

3.2 Sensitivity analysis of drawbeads

As discussed in previous section. After choosing the optimum material, the forming process is further optimized for draw bead strength. As shown in Fig.1:, there are around 20 draw beads segments around the die. To identify the critical draw beads, a sensitivity analysis is performed using 100 iterations. The CoP achieved for Crack and safe stretch is around 78% and 72.8% as highlighted in right end of Fig.21:1.The sensitivity analysis is expected to have a lower CoP(compared to 90%) as the material, one of the critical parameter is fixed during the run.

The results of sensitivity analysis are shown in below(a in Fig.21:).

- The draw beads(DB),2 and 3 seem to significantly affect the Safe stretch (above 10% effect, highlighted as green)
- DBS 12,13, 7 and 18 seem to be critical from Cracking perspective.

This is quite intuitive to understand once we see the FLD plots of the component under preliminary analysis(Fig.22:, a&b). There is some cracking observed at left and right edges and at bottom of the sheet. The drawbead closer to these regions seem to have significant impact on cracking and the drawbead at the top seem to have significant impact on the safe stretch. The insights from the sensitivity analysis are further used in the Forming plus Modal optimization analysis. The optimized design is shown in Fig.22:,c.

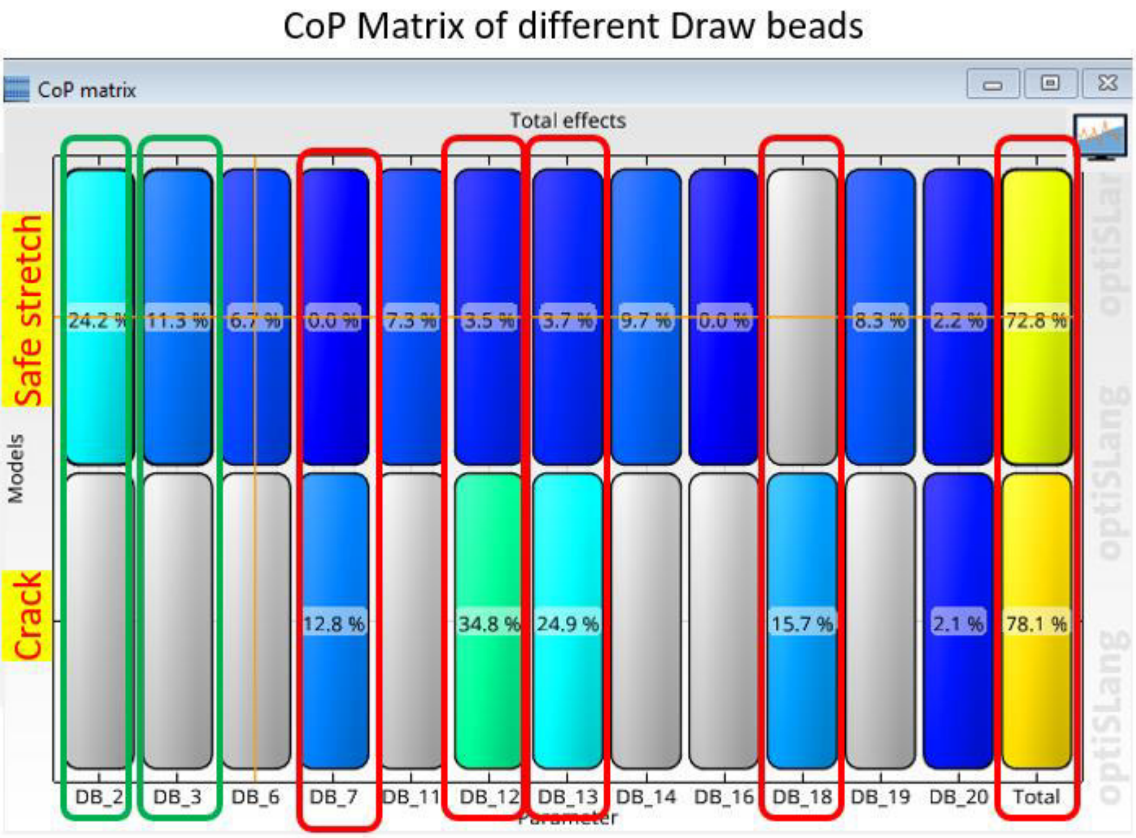
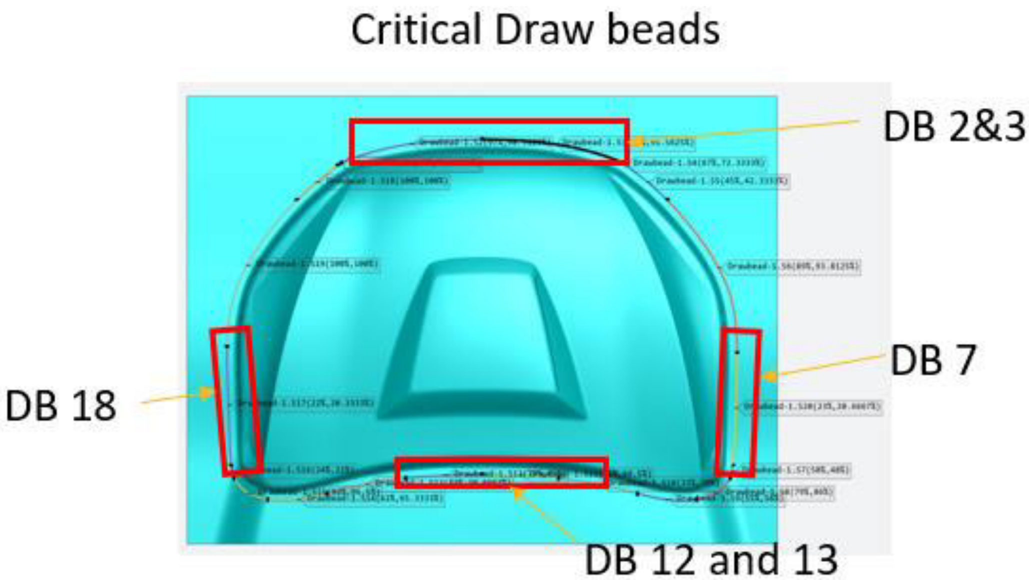
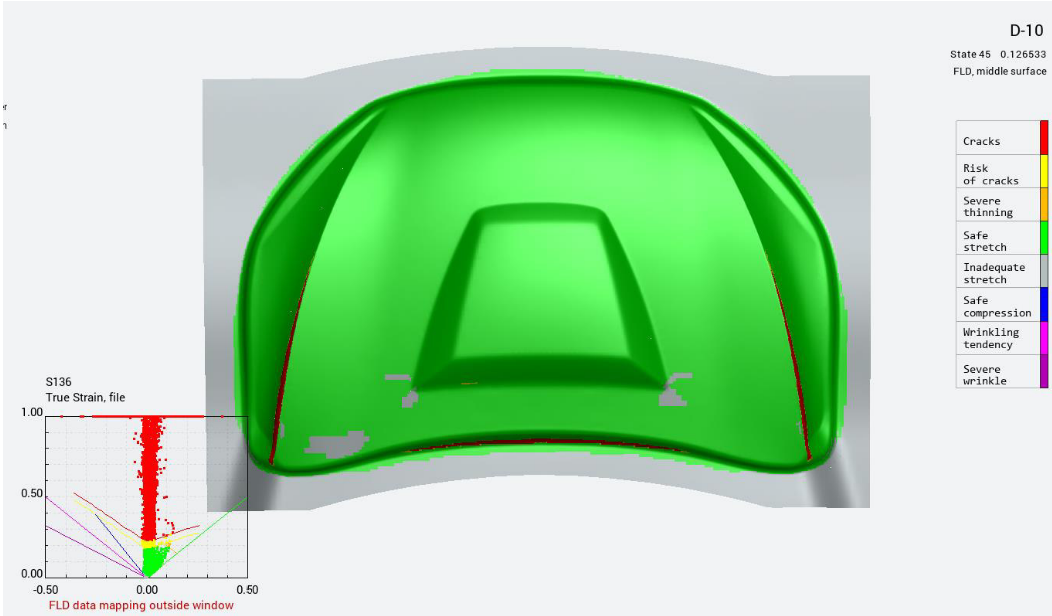


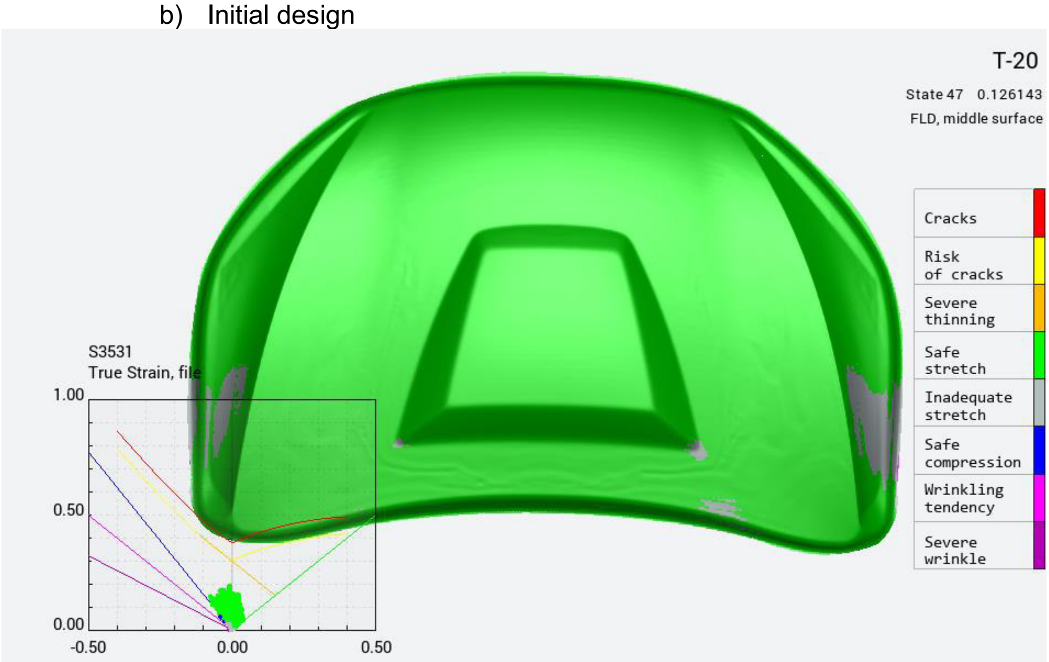
Fig.21: Drawbeads and their CoP as output by optislang



a) Critical Draw beads identified by Sensitivity analysis



b) Initial design



c) Optimized solution

Fig.22: A) Critical drawbeads B) initial design C) optimized design

3.3 Optimization of Forming plus Modal analysis

The optimal design of the Multi objective Forming plus Modal analysis can be chosen from the pareto front. The chosen optimal design has the optimum thickness to be 3.22 mm and the optimum % Restrain forces as shown in top right image of Fig.23:3. The optimum values are validated by inputting the optimum input parameters and rerunning the analysis. The comparison between values output from ROM and the actual output by simulations tabulated in Table 2:.Although there are some differences observed in the thickness values between ROM and optiSLang, the error seems to be within acceptable limit. Further as a corrective measure, more input parameters could be included to improve the COP of thickness.

The FLD plots in left top image in Fig.24:4. shows that the part is mostly in safe stretch without much crack indicating a feasible design. The mode shapes and the modal frequencies of the hood in shown in Fig.24:

Table 2: Comparison of ROM and Validation Run.

Output Quantity	ROM	Simulation
Frequency 1(Hz)	31.95	31.86
Frequency 2(Hz)	37.6	37.66
Min thickness	2.13	2.44

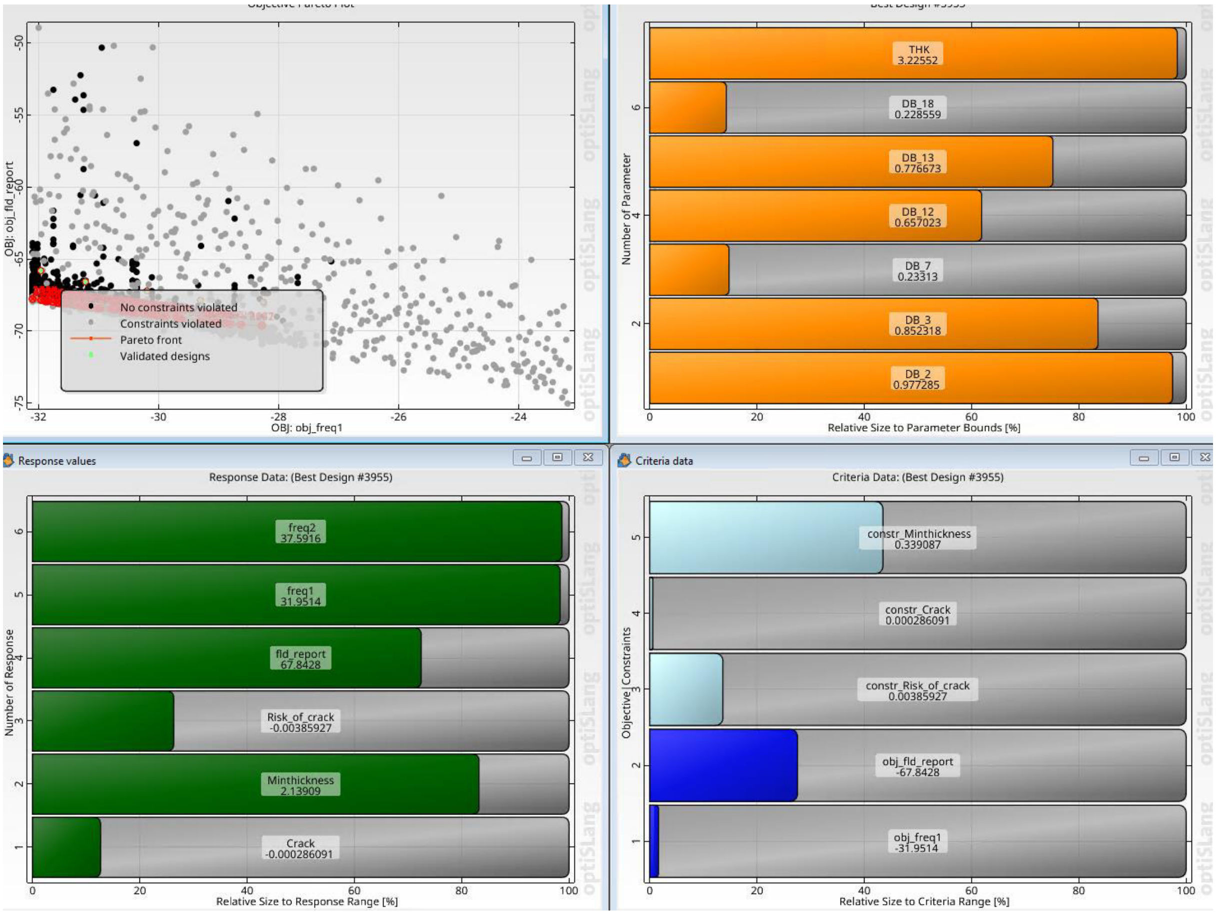
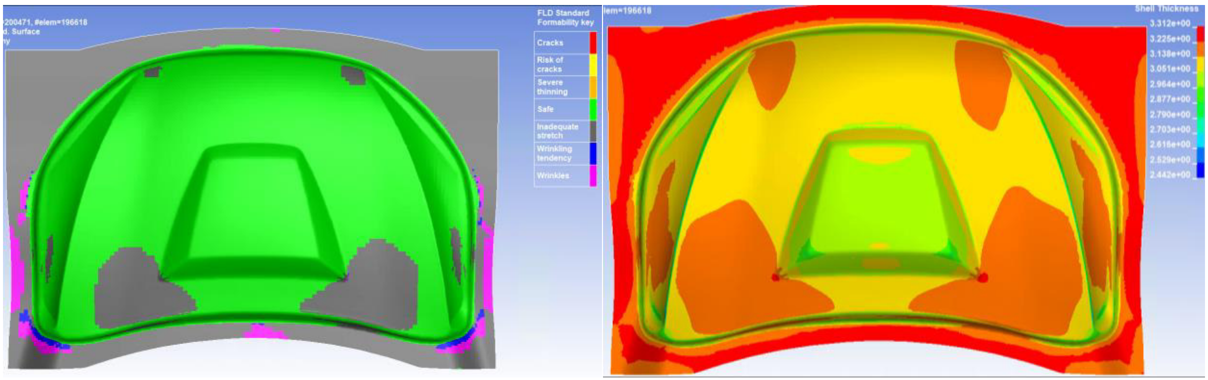


Fig.23: Output from optiSLang optimization



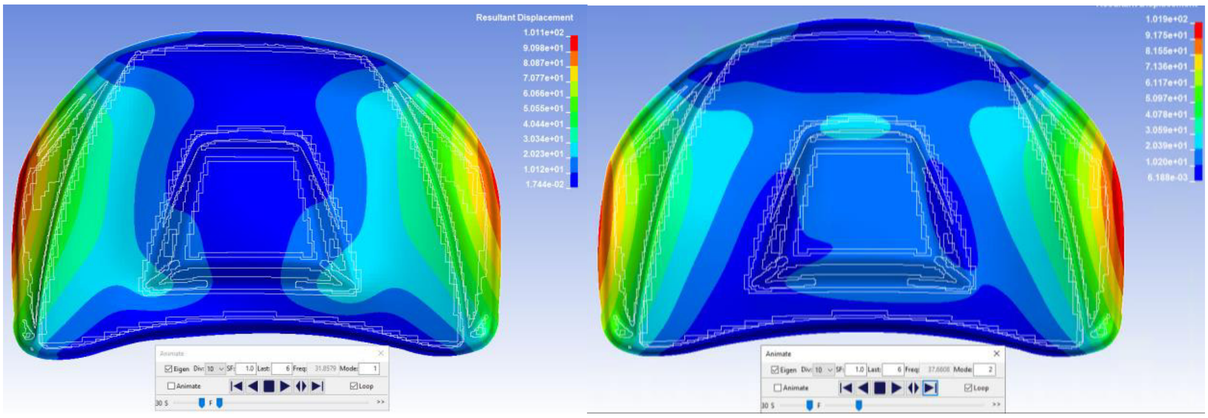


Fig.24: Results of validation run

3.4 Result transfer to mechanical

After optimization, user could transfer the optimized shape into next stage. In next stage, user often want to know the eigenfrequency and eigenmode to avoid resonance [7,8], improving NVH [9] and enhancing durability and safety [10]. To archive it, user could transfer the result from Ansys Forming to LS-DYNA, which could directly perform a modal analysis or transfer the data into Ansys Mechanical.

3.4.1 Transfer data to LS-DYNA

Because Ansys Forming is using LS-DYNA solver, there is no additional effort needed to use the forming result. When the Forming simulation is done, a dynain file is stored with thickness distribution and stress-strain data. This data will be used in modal analysis.
In LS-DYNA modal analysis, following control card is used:

```
*CONTROL_TERMINATION
$    ENDTIM    ENDCYC     DTMIN    ENDENG    ENDMAS
         1.
*CONTROL_IMPLICIT_GENERAL
$    IMFLAG       DTO    IMFORM      NSBS       IGS     CNSTN      FORM
          1      0.10         2         0         1         0         0
*CONTROL_IMPLICIT_EIGENVALUE
$#     neig    center     lflag    lftend     rflag    rhtend    eigmth    shfscl
       -706
$#   isolid     ibeam    ishell   itshell    mstres    evdump   mstrscl
                                                  1
```

In this case, modal analysis with considering prestress effects(IGS=1 in ***CONTROL_IMPLICIT_EIGENVALUE**). User also needs to define boundary conditions (***BOUNDARY_SPC_SET**) to fix the part (as shown in Fig.25:5) and perform a modal analysis.

Fig.25: Boundary conditions used for modal analysis

To compute eigenvalues intermittently , NEIG < 0 and IGS=1 and the time of eigen value extraction is specified as a negative value of curve defined using **`*DEFINE_CURVE`**.In the example, the implicit analysis is run for 1 second and the eigen values are calculated at the end of implicit run.Fig.26:6 shows the 2 eigenfrequencies and eigenmodes from LS-DYNA.

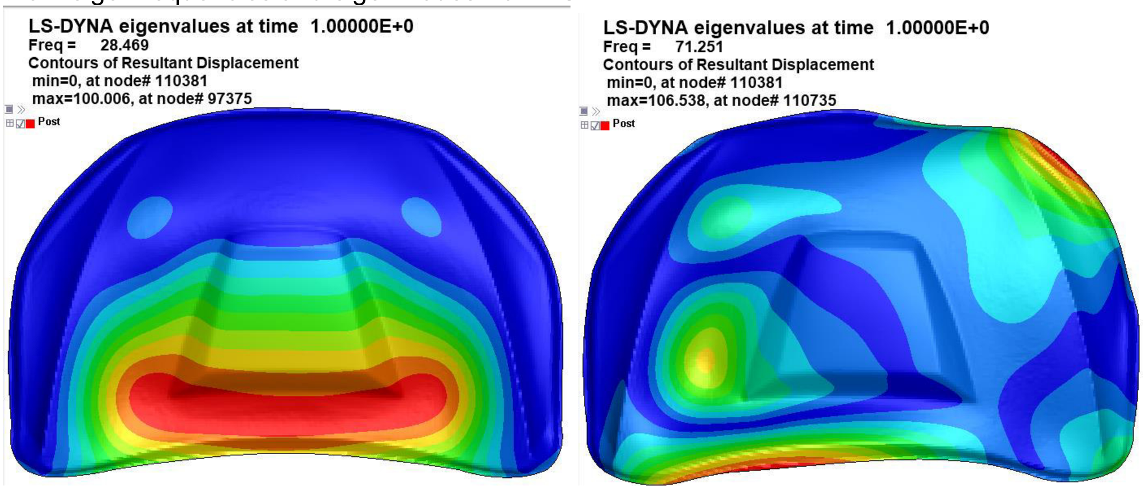


Fig.26: First 2 models of Card hood model under Bcs specified as shown in Fig.25:

As as part of implicit analysis during the intial part of the solution, the springback is automatically calcuated as shown in Fig.27:7

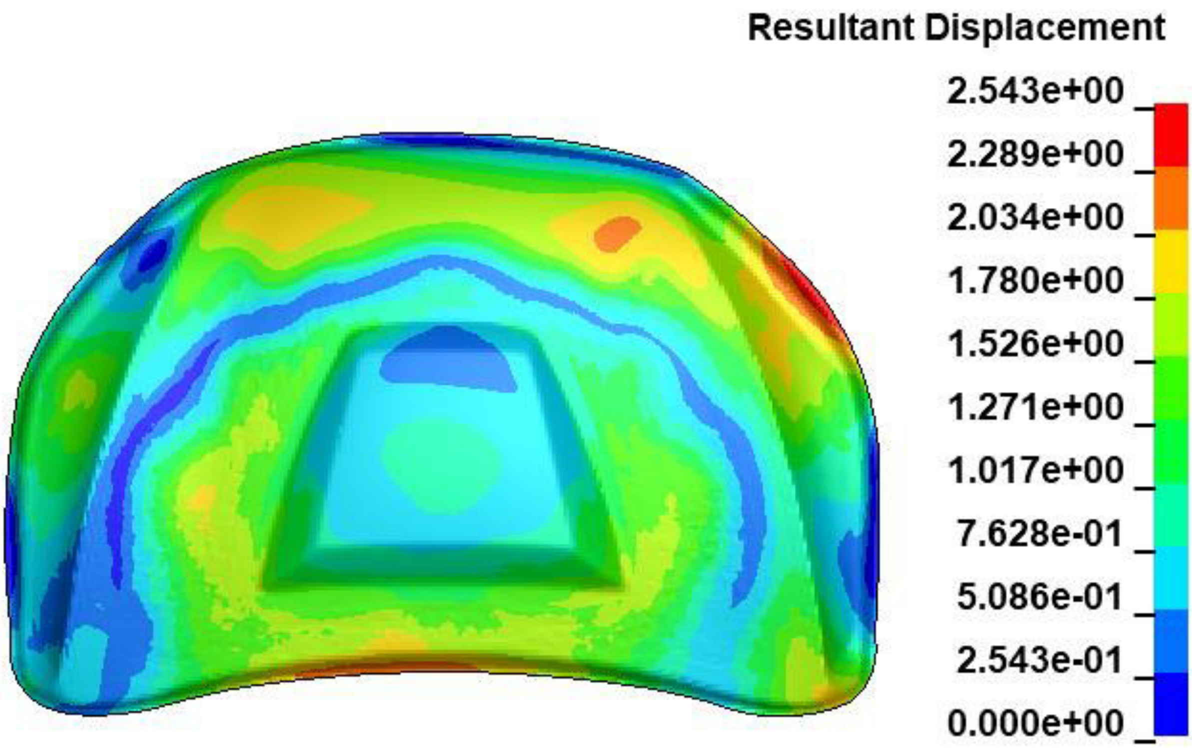


Fig.27: Springback results of LS-DYNA

3.4.2 Transfer data to Ansys Mechanical

To do the same as LS-DYNA in Mechanical Modal analysis takes more effort, because LS-DYNA and Mechanical do not use the same solver. Recommendations are:

- Using ELFORM=2 and NIP=5 in Ansys Forming
- Use initial fine mesh instead of mesh adaptivity

The Transfer workflow in Ansys Workbench is shown in Fig.28:8

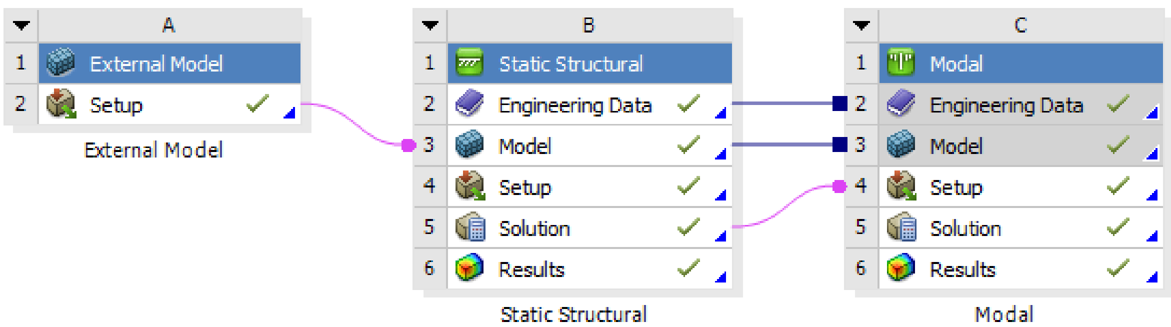


Fig.28: Workflow in Workbench mechanical

The dynain file could be imported into Ansys Workbench as external model. User needs to pay attention to the unit system in external model(shown in Fig.29:29). The default Workbench project is using SI unit (kg, m, s) but Ansys forming before uses Metric unit (tonne, mm, s).

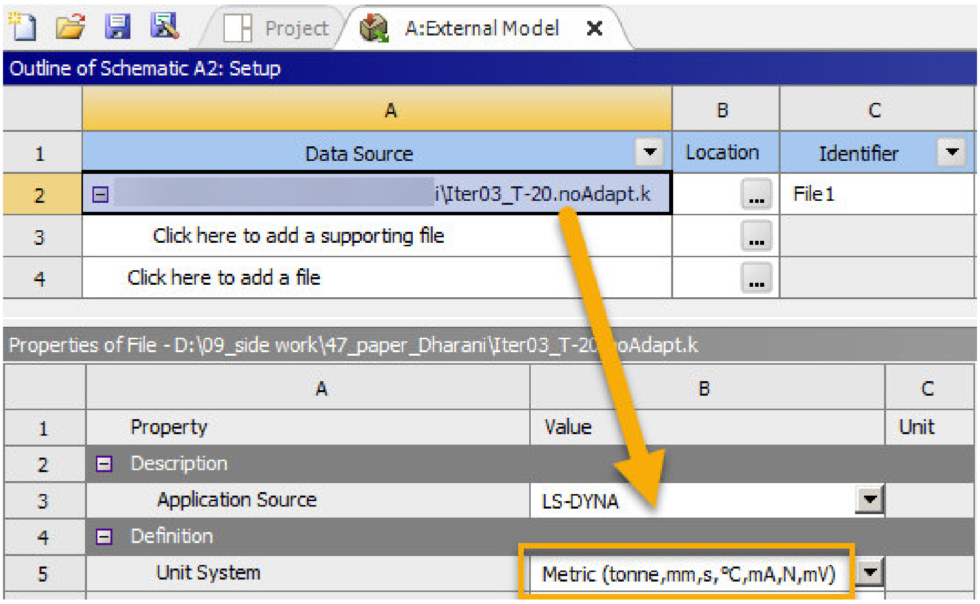


Fig.29: Settings in Ansys Workbench External Model

While importing, this unit system needs to be changed for the consistency.

The second step is connecting External Modal with static analysis as shown in Fig.28:. After importing the forming result, the shell thickness and initial stresses are used in static simulation as shown in Fig.30:.

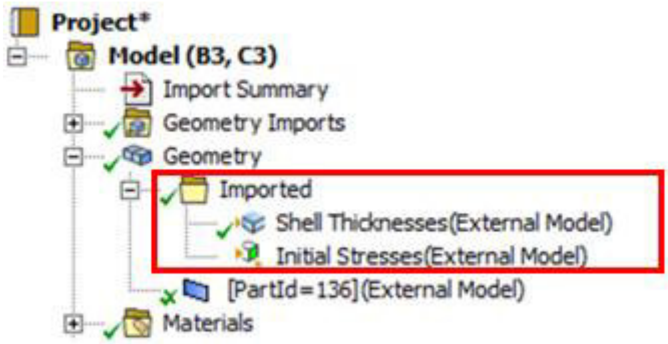


Fig.30: Shell thickness and stress import in Mehchanical

Then, the same nodes are fixed, and a static simulation is performed as a springback analysis. This is the same as in LS-DYNA approach in 3.4.1. with negative NEIG parameter. The springback form in Ansys Mechanical is showed in Fig.31:.

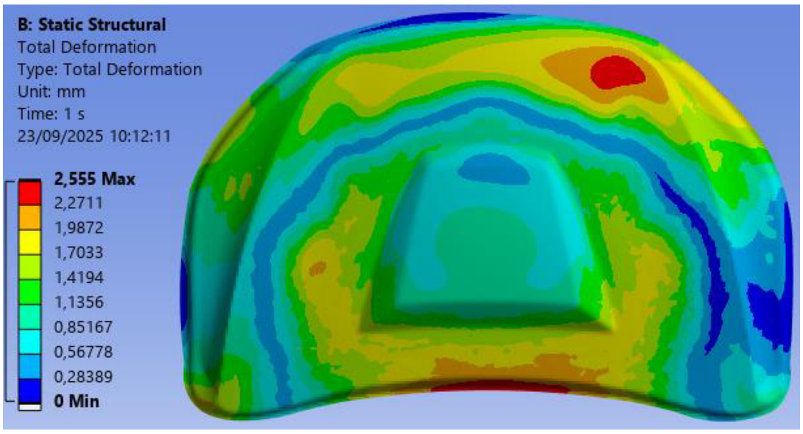


Fig.31: Springback analysis results from Ansys Mechanical

The next step is pre-stressed Modal analysis. The Modal analysis uses the prestress from the static solution(Solution of static structural is linked to setup of modal as shown in Fig.28:8) and calculated the eigenmodes show in Fig.32:. The results of LSDYNA and mechanical show good correlation and are tabulated in Table 3:

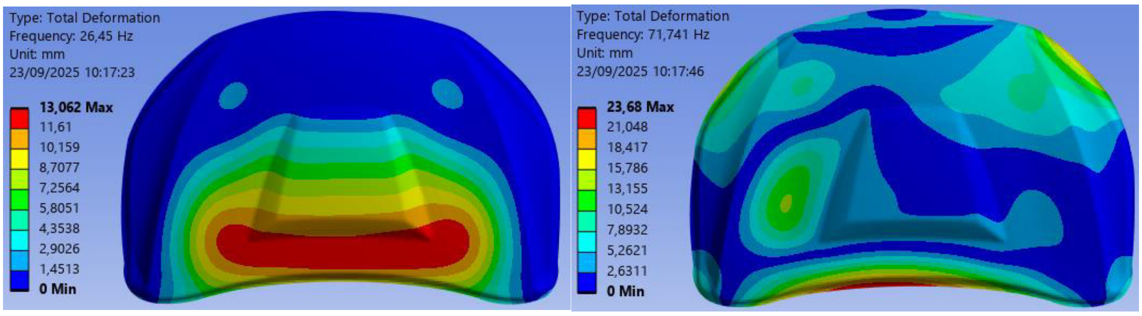


Fig.32: Modal analysis results from Ansys Mechanical

Table 3: Comparison of LS-DYNA and Mechanical results.

	LS-DYNA	Ansys Mechanical
Frequency 1(Hz)	28.47	26.45
Frequency 2(Hz)	71.25	71.74
Max springback(mm)	2.54	2.55

4 Summary

The study presented an integrated approach for material selection, forming process optimization, and downstream performance evaluation in sheet metal forming, using a car hood as a representative example. The proposed workflow leverages one-step forming simulations for rapid screening of candidate materials, enabling early-stage decisions that significantly impact manufacturability and product performance. The approach utilizes the information (FLD Report, thinning) from One Step simulations to screen group of candidate materials suitable for the geometry under consideration. Further selection of materials can be done based on well established guidelines from standard textbooks for choosing the materials.

Subsequent sensitivity analyses identified the most influential draw bead segments, allowing focused optimization of forming parameters. Furthermore, incorporating forming effects into modal analysis improves prediction accuracy of natural frequencies and ensured the final component and also the accuracy of downstream analysis like harmonic + fatigue calculations.

Overall, the workflow demonstrates that integrating forming simulation data early into the product development process enhances decision-making, reduces design iterations, and provides a more robust and manufacturable design. This methodology can be extended beyond hood panels to other automotive components where forming-induced effects significantly influence downstream structural performance – Denting analysis, Durability calculations, failure analysis of components.

Future work

The current python script is limited to extracting information from only `*MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC_NLP2_TITLE` and is demonstrated on a smaller dataset. The code could be extended to choose materials based on other criteria like weight, cost, corrosion resistance, conductivity etc.

Right now, the FLD properties of two materials are compared based on based on averaged ordinate values – major strain. The approach could be improved to compare the FLD properties of 2 materials more accurately.

5 References

[1] Chen, X., & Ghouati, O. Practical Optimization for Automotive Sheet Metal Components. *6th European LS-DYNA Conference.* Gothenburg, 2007

[2] Keshavamurthy, M., Zhang, J., Tang, Y., & Zhu, X. Highly Automated Springback Compensation of the Draw Die. *17th International LS-DYNA Conference 2024.* Plymouth, Michigan, 2024

[3] Gerlach, J., Kessler, L., Köhler, A.: "The forming limit curve as a measure of formability – Is an increase of testing necessary for robustness simulations?", ThyssenKrupp Steel Europe AG, Research and Development

[4] Clees, T., Steffes-lai, D., Helbig, M., Roll, K., Feucht, M, "Process Chain Forming to Crash: Efficient Stochastic Analysis," *European LS-DYNA Conference*, Issue F-IV-03, 2009, pp. 1–15. Available at: https://www.dynalook.com/conferences/european-conf-2009/F-IV-03.pdf

[5] *LS-DYNA Keyword User's Manual Volume I*, Version R15, Ansys Inc., 2024

[6] Altan, T., & Tekkaya, A. E. (2012). *Sheet metal forming: Fundamentals*. ASM International.

[7] Ryzhikov, V.A. Resonance Phenomena in Vehicle Suspension. In: Radionov, A.A., Gasiyarov, V.R. (eds) Proceedings of the 9th International Conference on Industrial Engineering. ICIE 2023. Lecture Notes in Mechanical Engineering. Springer, Cham. https://doi.org/10.1007/978-3-031-38126-3_63

[8] Reynders, E. System Identification Methods for (Operational) Modal Analysis: Review and Comparison. *Arch Computat Methods Eng* **19**, 51–124 (2012). https://doi.org/10.1007/s11831-012-9069-x

[9] Liu, Y., Liu, X., Dou, L., Luan, Y., Shi, L., Zheng, G. (2021). Sensitivity Analysis of NVH Structure of Car Body Based on Modal Analysis. In: Jain, L.C., Kountchev, R., Tai, Y. (eds) 3D Imaging Technologies—Multidimensional Signal Processing and Deep Learning. Smart Innovation, Systems and Technologies, vol 236. Springer, Singapore. https://doi.org/10.1007/978-981-16-3180-1_11

[10] Zhang, Z., Wu, C., Wu, Z., Lei, Y. (2021). Car Body Durability Analysis Based on Modal Superposition Method. In: Proceedings of China SAE Congress 2019: Selected Papers. Lecture Notes in Electrical Engineering, vol 646. Springer, Singapore. https://doi.org/10.1007/978-981-15-7945-5_8