User Manual – CAPY-Q: Statistical Process Analysis

Version: 2026.0.0

Table of Contents

1. Introduction

This manual provides instructions for using the user interface (UI) of CAPY-Q, a software tool developed for statistical analysis of FEM (Finite Element Method) based forming processes. The UI enables the definition and variation of process parameters for conducting **sensitivity studies**, **optimization** and **robustness analyses**.

2. General Overview

Menu Bar:

Access to project options (Load, Save, Save Project as, Send Project to & New Project) and program settings.

The top menu of CAPY-Q provides access to four main modules that guide the user through the process of setting up and evaluating statistical simulations. Each tab has a distinct role in the overall workflow:

Toolbar:

The CAPY-Q toolbar is a bar of icons used to select different CAPY-Q functions (Define project basics, controlling solver runs, create a statistical model and analyzing statistical quantities)

Modul 1: Setup

The Setup tab is the starting point of every analysis project. It includes the essential configurations for preparing a simulation.

Modul 2: Sensitivity Analysis

The Sensitivity tab allows you to evaluate how changes in input parameters influence the output of the forming process. Identify critical parameters that have the highest impact on key output responses. Use this tab to prioritize which parameters are most relevant and potentially optimize them later.

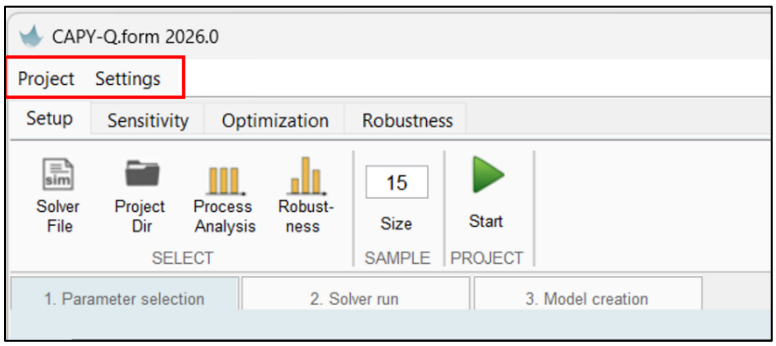
Modul 3: Optimization

The Optimization tab enables fine-tuning of input parameters to achieve specific target objectives (e.g. Failure, Thinning, Distance to Nominal, …). This module is ideal for improving process quality and identifying optimal process windows.

Modul 4: Robustness

The Robustness tab evaluates how stable the process is under normal distributed parameter variations by performing a Monte Carlo simulation. Outputs are the "Scrap-rate" and the CpK-Value (Process Capability Index).

3. Menu Bar



Functionality:

- Load and save a project
- Create new project
- Define settings

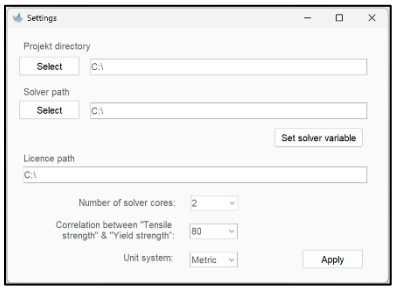
Key Components:

Project

Load…	Load a existing project (*.cqp).
Save	Save the project in the current project file.
Save project as	Save the project in a new project file, without adjusting the paths of already done simulation. *Note: It is recommended to use this function only to create a new storage file within the existing project directory to maintain a directory connection between the simulations and the project file.*
Send project to	Sends the project file, together with the underlying base simulation, to a selected path. The simulation paths are adjusted accordingly. *Note: This function is primarily intended for archiving completed projects. Only the basic simulation and the project file are moved. The design simulations are not moved, as these can be regenerated if necessary.*
Send stage to	Each time a process stage is imported, it is automatically saved independently in the project file and can be accessed individually. This function allows a process stage to be transferred to another project file.
Delete stage	Each time a process step is imported, it is automatically saved independently in the project file and can be accessed individually. This function allows process steps that are no longer required to be deleted from the project file.
New project	Create a new project. All panels and results will be reset.

Settings

Settings

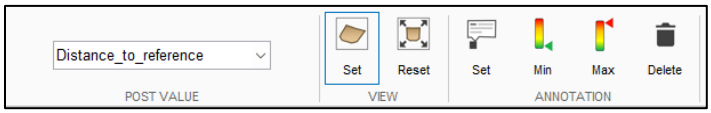


In the “Settings” panel, you can save default settings that will be applied to every new project.

This allows you to preset the “Project directory,” the location of the solver executable (“Solver path”), the solver variables (if available for your simulation application), the number of solver cores, the input correlation between “Tensile strength” and “Yield strength,” and the unit system.

The project directory defined in this view serves as the starting path for all searches and for creating new projects. If it remains unused or the selected path is deleted, each path selection will default to starting at C:\.

4. Result visualization



Functionality:

- Selecting simulation results (post value)
- Controlling the view and annotations
- Create model and analyzing statistical quantities.

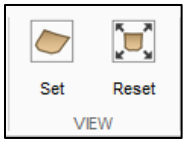
Key Components:

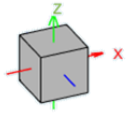
Note: Toolbar elements that are available in all modules are **POST VALUE**, **VIEW** *and* **ANNOTATION**. *All other toolbar elements are module-specific and are explained in the respective module chapters.*

Post Value Drop Down

Here, you can select the various simulation results for visualization. The selected post size is displayed as a surface plot on the component geometry.

View



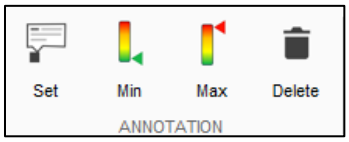


The **View** buttons are used to select a user-specific view. The **Set** button saves the current view, and the **Reset** button restores the saved view.

Note: The saved view is applied to all modules.

Another way to set axis-specific views is to use the axis cube at the bottom right of the UI. Clicking on the cube faces allows you to select X, Y, and Z views.

Annotation



The **Set** button can be used to place a annotation (data tip) on the mesh surface. To do this, move the mouse pointer to the desired location on the mesh and press the left mouse button.

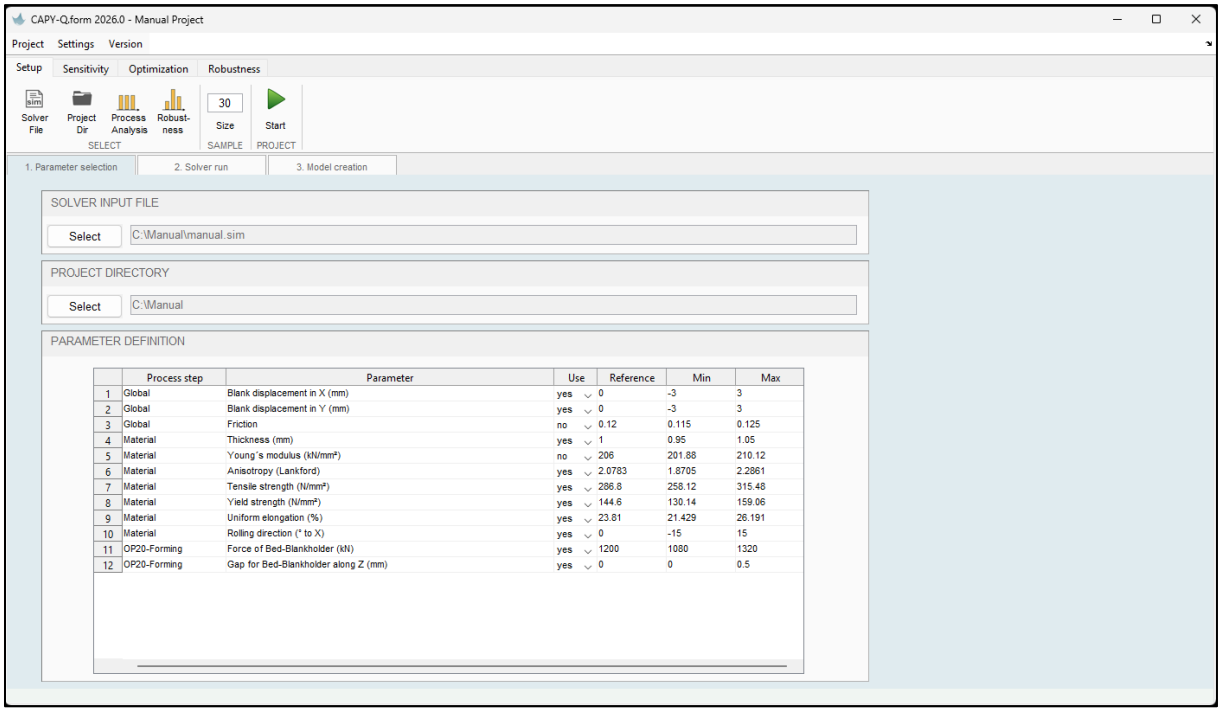
Note: Once the Set button has been pressed, you are in data tip mode. To exit, press ESC or the Delete button.

The **Min** and **Max** buttons automatically set an annotation on the mesh at the minimum or maximum value. To do this, the results of all FE nodes in the mesh are compared and the minimum or maximum result value of all FE nodes is displayed.

The **Delete** button deletes all annotations on the mesh surface.

5. Module 1: Setup / Parameter selection

Use this tab to ensure that the process remains reliable even with material fluctuations or production variability.



	Process step	Parameter	Use	Reference	Min	Max
1	Global	Blank displacement in X (mm)	yes	0	-3	3
2	Global	Blank displacement in Y (mm)	yes	0	-3	3
3	Global	Friction	no	0.12	0.115	0.125
4	Material	Thickness (mm)	yes	1	0.95	1.05
5	Material	Young's modulus (kN/mm²)	no	206	201.88	210.12
6	Material	Anisotropy (Lankford)	yes	2.0783	1.8705	2.2861
7	Material	Tensile strength (N/mm²)	yes	286.8	258.12	315.48
8	Material	Yield strength (N/mm²)	yes	144.6	130.14	159.06
9	Material	Uniform elongation (%)	yes	23.81	21.429	26.191
10	Material	Rolling direction (° to X)	yes	0	-15	15
11	OP20-Forming	Force of Bed-Blankholder (kN)	yes	1200	1080	1320
12	OP20-Forming	Gap for Bed-Blankholder along Z (mm)	yes	0	0	0.5

Functionality:

- Define project basics
- Activate parameters and define range (Min/Max)

Key Components:

Solver Input File

Allows selection of the solver file containing the data of the reference simulation (see Workflow - Step 1).

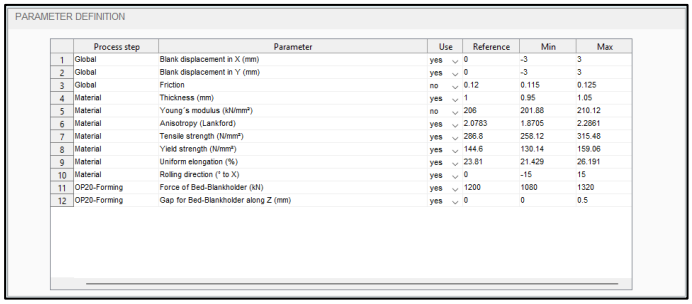
Note: You can either use the button "Solver File" in the toolbar or use the "Select" button in the "Solver Input File" panel. Both buttons have the same function.

Project Directory

Path where simulation results and temporary files are stored (see Workflow - Step 2).

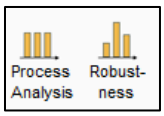
Note: You can either use the button "Project Dir" in the toolbar or use the "Select" button in the "Project Directory" panel. Both buttons have the same function.

Parameter Definition Table Lists all modifiable parameters (see Workflow - Step 3).



	Process step	Parameter	Use	Reference	Min	Max
1	Global	Blank displacement in X (mm)	yes	0	-3	3
2	Global	Blank displacement in Y (mm)	yes	0	-3	3
3	Global	Friction	no	0.12	0.115	0.125
4	Material	Thickness (mm)	yes	1	0.95	1.05
5	Material	Young´s modulus (kN/mm²)	no	206	201.88	210.12
6	Material	Anisotropy (Lankford)	yes	2.0783	1.8705	2.2861
7	Material	Tensile strength (N/mm²)	yes	286.8	258.12	315.48
8	Material	Yield strength (N/mm²)	yes	144.6	130.14	159.06
9	Material	Uniform elongation (%)	yes	23.81	21.429	26.191
10	Material	Rolling direction (° to X)	yes	0	-15	15
11	OP20-Forming	Force of Bed-Blankholder (kN)	yes	1200	1080	1320
12	OP20-Forming	Gap for Bed-Blankholder along Z (mm)	yes	0	0	0.5

Parameter selection



After importing the solver input file, all process or robustness parameters can be selected in the parameter table using these two buttons. These are set to yes in the "Use" column. A standardized preselection is made, whereby "Process Analysis" selects parameters that can be adjusted during the forming process (forces, stroke). In "Robustness," parameters are selected that cannot be controlled by the process operator, or only with great effort (e.g., material properties).

Sample size

Input field where the number of simulation runs (also called samples or designs) can be set.

Note: The optimal sample size is determined automatically. It should only be modified by experienced users and/or in consultation with support.

Start button

Start the project (see Workflow - Step 4).

Workflow:

Step 1: Select the solver input file (*.dat /*.pc) of a reference simulation. It is recommended that this simulation has already been calculated once to ensure that it is calculated reliably.

Step 2: Select a project directory. The variant calculations are carried out in this directory and the project results are saved. It should be noted that the directory should not be moved until the calculation of the variant simulations has been completed.

Step 3: The "Parameter Definition" section allows you to activate specific process parameters and set their value ranges for statistical evaluation. This was automatically identified in the solver input file.

Column	Description
Process step	Process step (e.g., Material, OP10: Blank Fitting)
Parameter	Name of the parameter and unit system
Use	Defines whether the parameter should be included in the statistical analysis (“yes”) or not (“no”).
Reference	This is the nominal value from the reference simulation. It cannot be changed, ensuring that the reference simulation can always be calculated successfully.
Min	Minimum value for variation, this must always be less than or equal to the reference value
Max	Minimum value for variation, this must always be greater than or equal to the reference value

Notes:

1. Some material cards and curves must be derived or created from the original simulation file. This may lead to slight numerical differences between the selected simulation and the same simulation calculated in CAPY-Q. To keep the CAPY-Q project internally consistent and maintain high model quality and reliability, a reference simulation (Design0000) preprocessed by CAPY-Q is performed again.
2. Use constraints to respect physical or manufacturing limits.
3. Mathematical modelling can be less accurate at the parameter limits. It is therefore recommended to specify the limits a little larger than those to be considered later in the optimization or robustness analysis.
Example: If the ‘uniform elongation’ can vary between 23% and 25% according to the supplier's data sheet. The analysis should be carried out for a range between 22.5% and 25.5%.
Enlarging the parameter space moderately does not lead to a deterioration of the model and does not increase the number of samples required. As long as you respect physical limits.

Parameters:

Parameter	Description
Blank displacement	Varies the insertion position of the blank in the first process step. The variation is based on the coordinate system defined for this first process step and is defined independently for the X and Y directions. The resulting vector for simultaneous variation is calculated automatically when the simulations are set up. *Note: To use this parameter, you should make sure that any guides modeled in the simulation allow a variation in the specified range at all. Otherwise, there may be intersections and incorrect results.*
Friction	All friction coefficients in the respective design of the simulation model are varied identically, regardless of whether they differ in reference simulation. *Note: Friction has a major influence on the results of forming simulations. At the same time, it is a production parameter that is very difficult to adjust or measure. It is recommended that you only manipulate*

	this parameter if you want to test your tooling method specifically for this challenge.
Friction Curve	This parameter is not offered by every simulation/solver version. If the imported reference model contains pressure-dependent friction coefficient models, the curve can be adapted by adjusting the exponent (drop in the curve) and the coefficients at the reference pressure value point (shifting the curve). *Note: Friction has a major influence on the results of forming simulations. At the same time, it is a production parameter that is very difficult to adjust or measure. It is recommended that you only manipulate this parameter if you want to test your tooling method specifically for this challenge.*
Thickness	Thickness in mm or inch
Young's Modulus	The modulus of elasticity essentially influences the numerical springback behaviour of the deep-drawn part. You should only adjust this if your material shows a variation according to the supplier data sheet.
Yield Strength & Tensile Strength	There is a fixed relationship (correlation) between the yield strength and tensile strength parameters. This must be considered in both parameters in order to achieve realistic results. For this reason, the parameters can only ever be varied together. Individual selection prevents the project from being started. *Note: When setting up the variant calculation, a correlation of 80 % between the parameters is assumed based on experience. This value can be adjusted in the 'Setting' menu. This option is only recommended for users with in-depth material knowledge.*
Uniform elongation	The variation of the uniform elongation can be adjusted according to the supplier data sheet.
Rolling direction	With an anisotopic material model, the rolling direction can be varied depending on the coordinate system of the first process step. The reference vector X or Y is selected according to the smallest angular difference in the reference simulation; at 0°, the X vector is always selected.
Anisotropy	According to Lankford, the anisotopy (r0, r45, r90) is summarized in the resulting value rm and written back by the fixed ratio equation when the designs are created.
Height of beads	The height is adjusted on the basis of the Z vector of the coordinate system of the respective drawing step. Note: The height of the drawbeads can only be manipulated if this has been prepared accordingly in the UI of the solver used. **To do this, it is necessary for the naming of the tool parts to begin with "Beads_".**
Distance / Gap	Depending on the solver used, both Distance and Gap describe the distance between the listed tools. The value is added to the sheet thickness in the reference

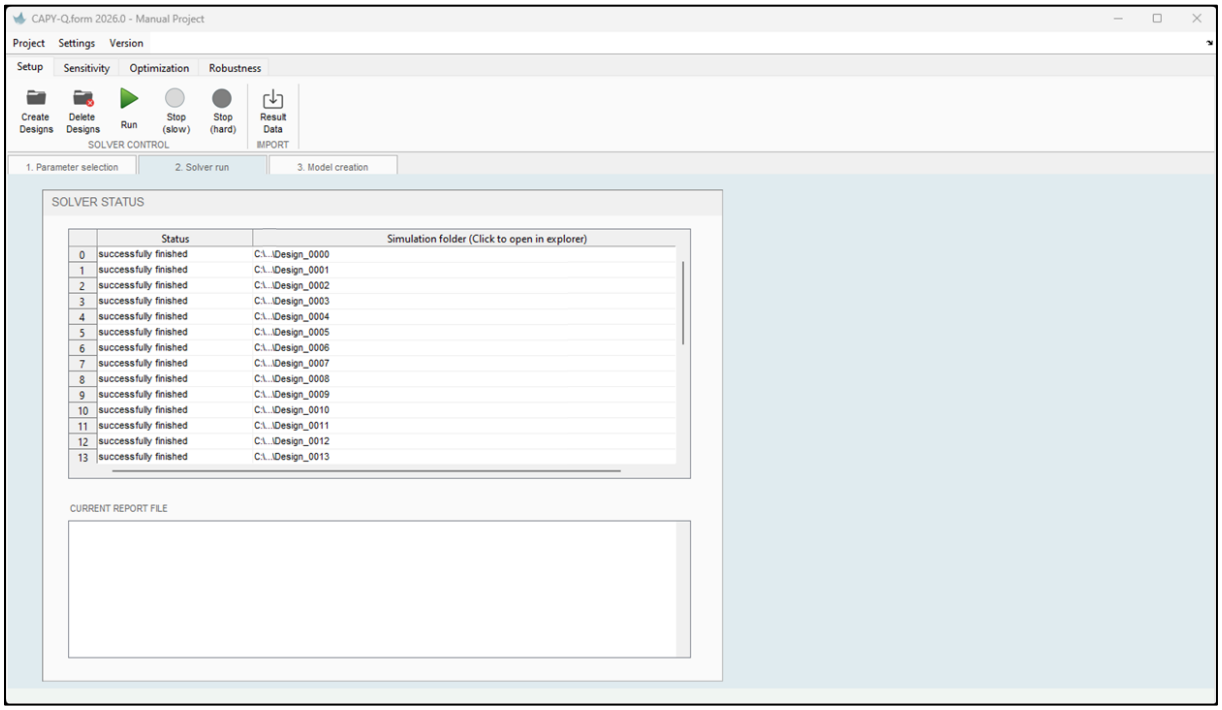
	simulation as well as in the variant simulation. Sheet thickness variations are also considered, so that parameterization does not result in the blank being crushed. The minimum must always be greater than 0.
Force	Forces are always associated with a specific tool. If a force is assigned to a particular blank holder, it is applied both when closing the blank holder and during the subsequent process.

By using the Process Analysis or Robustness buttons, you can simplify the selection of suitable variables for the respective type of analysis. The recommended parameters are selected automatically.

Depending on the number of selected parameters, the size of necessary calculations is adjusted automatically. The number (sample size) can be manipulated by experienced users. However, increasing the number generally does not improve the quality of the model (see sensitivity analysis), whereas the quality of the model can quickly deteriorate if the number of calculated parameters is reduced. The size should never be below 10.

Step 4: Pressing "Start" generates the Design of Experiment (DoE) based on defined parameters, creates the project file in the project directory and saves the entries made so far for the first time.

6. Module 1: Setup / Solver run



Functionality:

- Creating the designs
- Management and control of solver runs
- Importing the result files

Key Components:

Solver Control

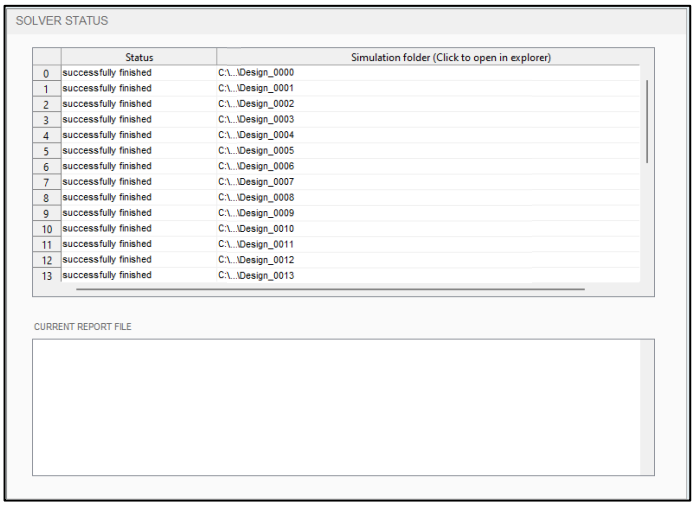
The **Solver Control** toolbar allows you to create / delete the design folders. You can also start and stop the simulation (see Workflow - Step 1 and 2).

Result Data

After simulation, this imports output results into the analysis environment (see Workflow - Step 3).

After pressing the button, the "Result File Import" panel opens. Here, the user selects the reference to use and chooses the result files from the desired stage to load.

Simulation Status Displays the progress of each design simulation. Includes the status (e.g., "successfully finished") and simulation folder. It also shows logs or reports associated with the selected simulation.



Workflow:

Step 1: Create the design folders. The command files for the solver are created at the same time. These include the simulation path, the solver path and the number of CPU cores available.

Step 2: Run the simulations. The simulations are processed chronologically; a parallel calculation is not carried out.

The "Stop slow" function gives you the option of interrupting the solver process. In this case, the current design is calculated until the end of the current simulation and then no further simulations are started.

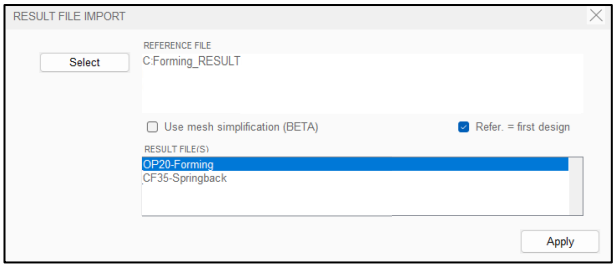
The "Stop hard" function immediately interrupts ALL simulations running on the respective PC, regardless of whether they were started in CAPY-Q.

The simulation can be resumed at any time. However, simulations that have been stopped hard must be recalculated from the beginning. If you select a project directory, the variant calculations are carried out in this directory and the project results are saved. It should be noted that the directory should not be moved until the calculation of the variant simulation has been completed.

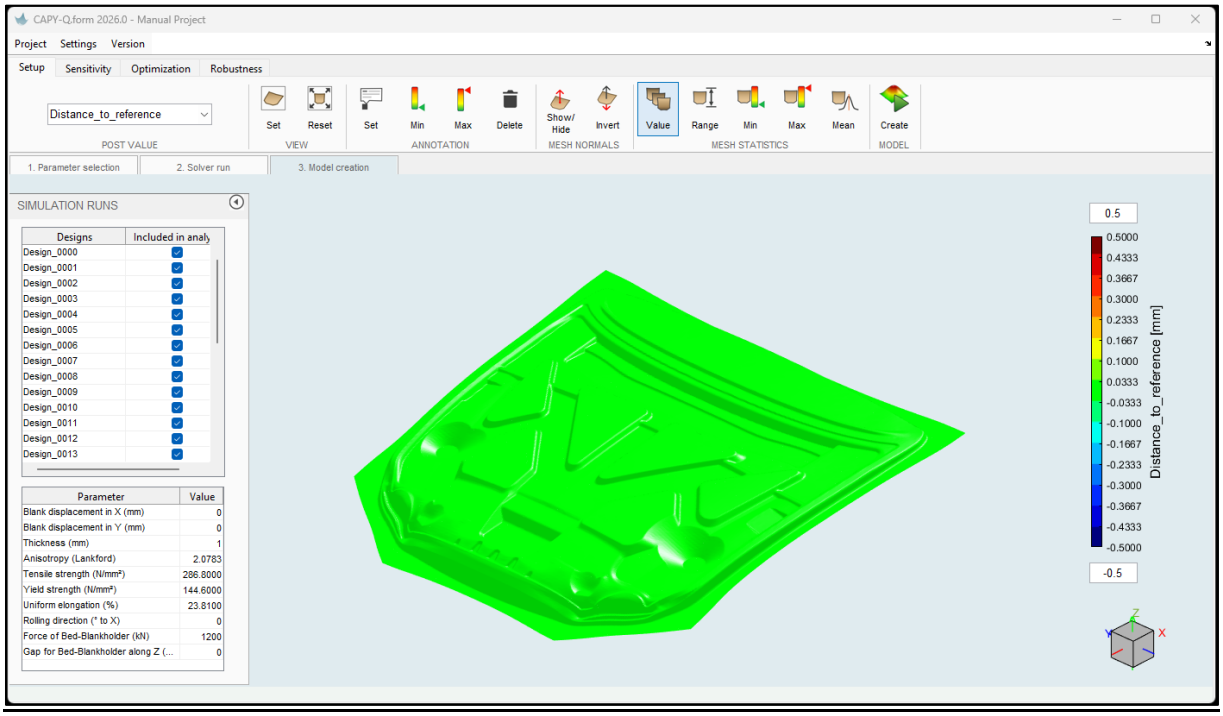
Step 3: Import Result Data by pressing the **Result Data** button. The **Result File Import** panel with all available result files of the current simulation is displayed. You can evaluate one process stage in each project. For this reason, you have the option of saving the project file under a new name when loading a new/additional project stage. In this case, there are several independent project files for the respective process steps in your project directory.

When the checkbox “Refer. = first design” is selected, the first simulation run (Design0000) is automatically used as a reference. Otherwise, the user must define their own reference mesh (usually CAD-0) in STL format.

The “Use mesh simplification” (only available for BM-S Solver) checkbox can be used for large simulation meshes. This will attempt to reduce the number of FE nodes to simplify the mesh. *Please note: This is a BETA version.*



7. Module 1: Setup / Model creation

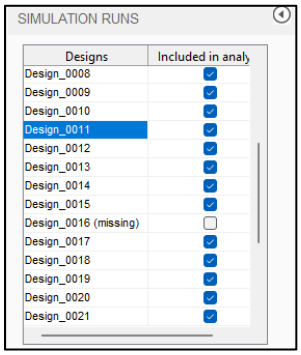


Functionality:

- Checking imported designs (simulation runs) for errors
- Comparison of the different simulation variants with each other for different simulation results
- Creating a new mathematical sensitivity and robustness model based on the selected designs

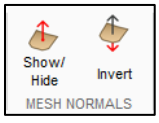
Key Components:

Simulation Runs Panel



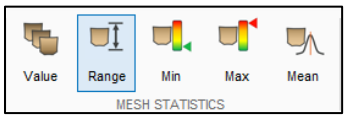
The panel allows you to switch between the calculated designs (simulation runs) for a selected simulation result (distance to reference, failure, …). The results are displayed individually for each design with the associated values of the parameters. By checking the box in the table column "included in analysis," you can manually select designs that should or should not be included in the model analysis. Designs that were not successfully calculated (missing) are also displayed (see Workflow - Step 1).

Mesh Normals



With the **Mesh Normals** Buttons the principal normal direction of the reference mesh can be shown and inverted in sheet thickness direction.

Mesh Statistics



Five statistical values can be displayed using the "Mesh Statistics" buttons.

When the **"Value"** button is selected, the currently selected simulation run results in the **Simulation Runs Panel** is displayed as a color surface plot.

All of the following buttons are design independent.

Button **"Range"** - The range of variation between the maximum and minimum values across all simulation runs (designs) is displayed at each FE node of the formed part as a color surface plot (see Workflow - Step 1 & 2).

Button **"Min"** - The minimum value across all simulation runs (designs) is displayed at each FE node of the formed part as a color surface plot.

Button **"Max"** - The maximum value across all simulation runs (designs) is displayed at each FE nod of the formed part as a color surface plot.

Button **"Mean"** - The mean value across all simulation runs (designs) is displayed at each FE node of the formed part as a color surface plot.

Model Create

Button for calculating a new mathematical model for robustness and sensitivity analysis. The model is calculated for **all** simulation result variables (post values) and takes approximately 1 to 2 minutes to calculate (see Workflow-Step 3).

Workflow:

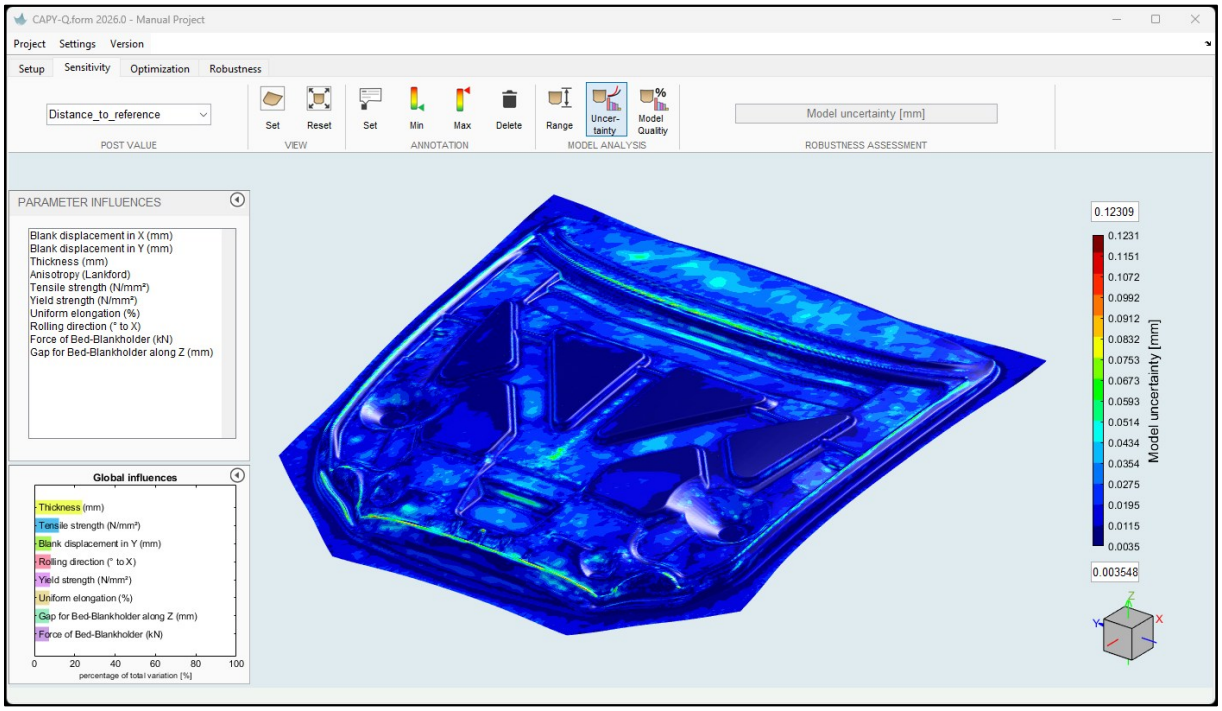
Step 1: In the first step, it is important to look at the various simulation results in the **Simulation Runs** panel to see whether all designs have been run correctly.

If errors are found in the calculation, the designs can be removed from the model calculation by removing the check mark in the Simulation Runs Panel.

Step 2: In the next step it is also possible to start an initial statistical pre-analysis. For example, the range (variation across all simulation variants at each FE node of the component) in the **Mesh Statistics** toolbar can be viewed. Areas with high variation range can be identified here. These are the areas that are particularly relevant for the robustness analysis.

Step 3: After the preliminary analysis, the model is created in the third step by pressing the **Model Create** button. The model is then automatically created based on the selected designs.

8. Module 2: Sensitivity Analysis

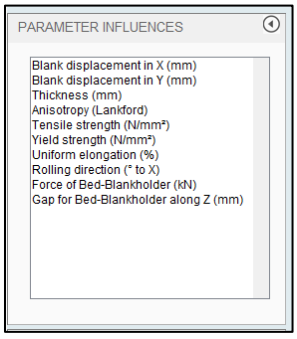


Functionality:

- Show global/ local parameter influences for a selected simulation result (post value).
- Show the model uncertainty and the model quality.

Key Components:

Parameter Influences

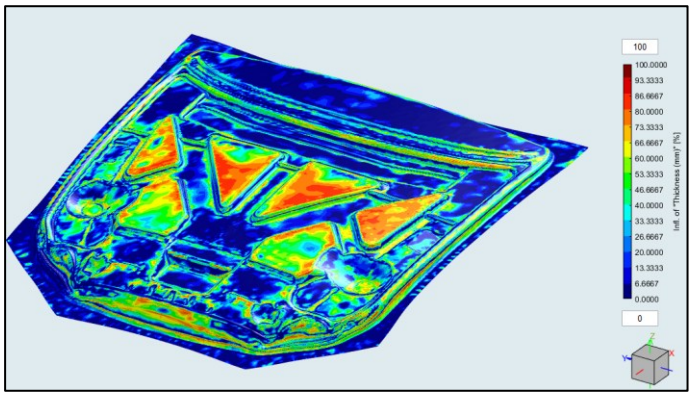


The panel allows you to switch between the different input parameters and show the influence (sensitivity) of each parameter on the selected simulation results (e.g. distance to reference) as a 3D surface plot.

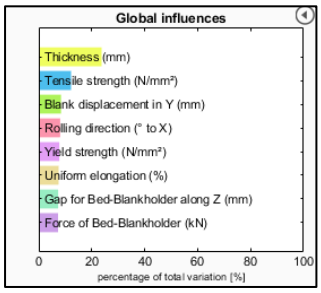
The influences can vary between 0 % and 100 %.

0 % means that the input parameter has no influence (effect) on the selected simulation result.

A high percentage value means that the input parameter strongly influences the simulation result and that a change in the result can be achieved via this parameter.



Panel of bar chart



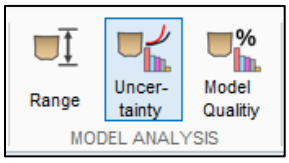
The bar chart shows the percentage influence of each input parameter on the currently selected post value (simulation result). The longer the bar, the greater the influence.

A distinction can be made between global and local influence.

The **global influence** indicates which parameter has the greatest "global" effect on the selected simulation result across all FE nodes of the formed part.

The **local influence**, on the other hand, shows the effect of each input parameter on a specific FE node of the formed part. If a new annotation is placed on a specific FE node of the formed part, the bar chart display changes from "global" to "local" influences (see Workflow - Step 2).

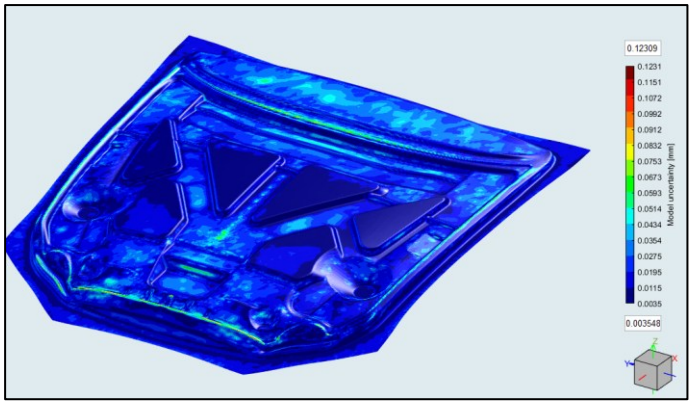
Model uncertainty



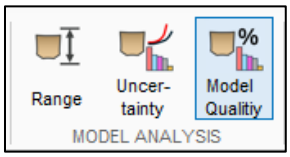
Button to display absolute model uncertainty. Model uncertainty indicates the **mean error** of the model at each FE node of the formed part. The mean error of the model indicates how well or poorly the model can predict the selected simulation result (post value).

The unit of model uncertainty is always related to the selected simulation result (post value). For example, if the post value selected is "Distance to reference", the model uncertainty is usually in millimeters mm or inch.

A model uncertainty of 0.01 mm means that the post value "Distance to reference" is predicted with an uncertainty of +- 0.01 mm. The higher the model uncertainty, the worse the model prediction (see Workflow - Step 1).

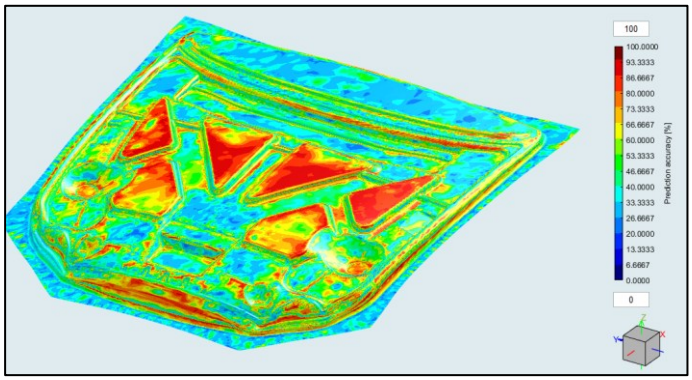


Model Quality



The model quality shows the cross-validated R^2 value for each node of the formed part. The R^2 can take values between 0 % and 100 %. 0 % indicates weak predictive ability and 100 % indicates perfect predictive ability of the model.

However, it must be considered that the R^2 value is a relative value that always refers to the range (distance between the smallest and largest value, individually at each node) of variation. This can lead to the R^2 being poor for small ranges, even though the model uncertainty is very small. **It is therefore always better to take Model uncertainty into account when evaluating the model.**



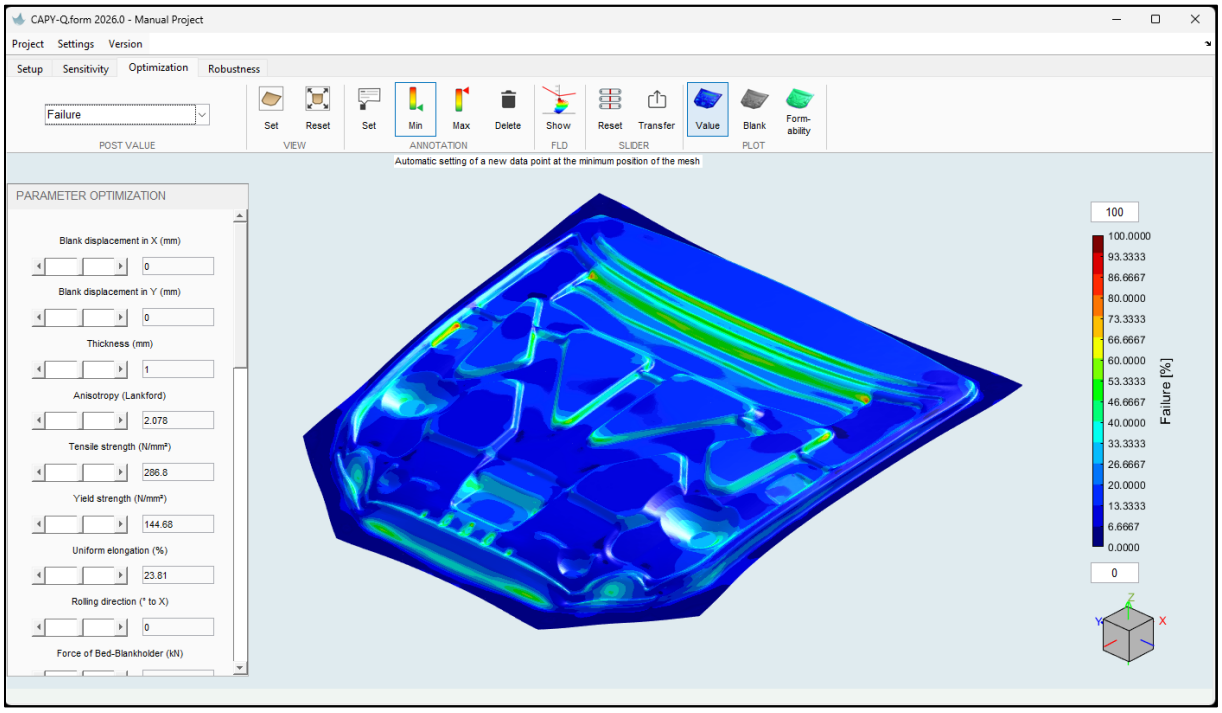
Workflow:

Step 1: In the first step, it makes sense to look at the **Model uncertainty** to ensure that the model has sufficient predictive capability. If the model uncertainty is too high, check again in "**Module 1 - Model creation**" to see whether all simulation variants have been calculated correctly.

Increasing the number of samples required for model creation generally rarely leads to a significant improvement in model quality. Before taking this step, we recommend contacting support to review the simulation model and the designs that have already been calculated.

Step 2: Next, the influences (sensitivities) of the parameters can be analyzed by the **Panel of bar chart**. To do this, you can display the local influences of all input parameters simultaneously by setting an annotation. Alternatively, you can select the influences of each parameter individually in the **Parameter Influences** panel. The goal here is to find those input parameters that have a particularly large influence (effect) in the critical component areas (areas with large variation).

9. Module 3: Optimization

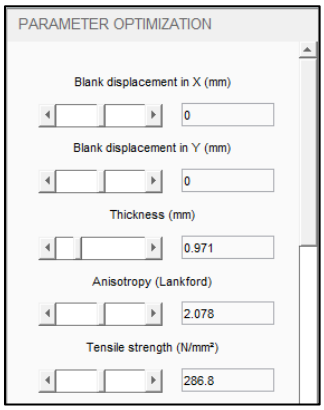


Functionality:

- Optimization of the simulation results by sliders

Key Components:

Parameter Optimization



The input parameters can be varied within the parameter limits using sliders. The resulting outcome, which is selected via the "Post Value" drop-down menu, is calculated in real time and displayed using a surface plot.

Plot Buttons

There are three different display options for the surface plot: "Value," "Blank" and "Formability"

With the **"Value"** button the currently selected simulation result (post value) is displayed as a color plot.

"Blank" displays the actual deformation of the formed part. However, this button can only be selected for the post value "Distance_to_reference." The deformation of the formed part

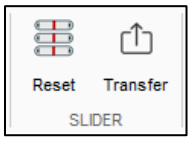
is always relative to the selected reference mesh you have selected at the result data import.

The **"Formability"** button shows the predicted formability map directly on the part.

FLD Button

The FLD **"Show"** button displays the approximate forming limit diagram for the current slider selection in a separate window.

Slider Buttons



The **"Reset"** button resets the slider values to their initial values (Design 0).

Once an optimal design has been found, the slider values can be transferred to the robustness analysis using the **"Transfer"** button. It should be noted that only adjustable values (forces, distances, blank displacement) are transferred, as these are the non-scattering parameters that can be directly adjusted (see Workflow - Step 2).

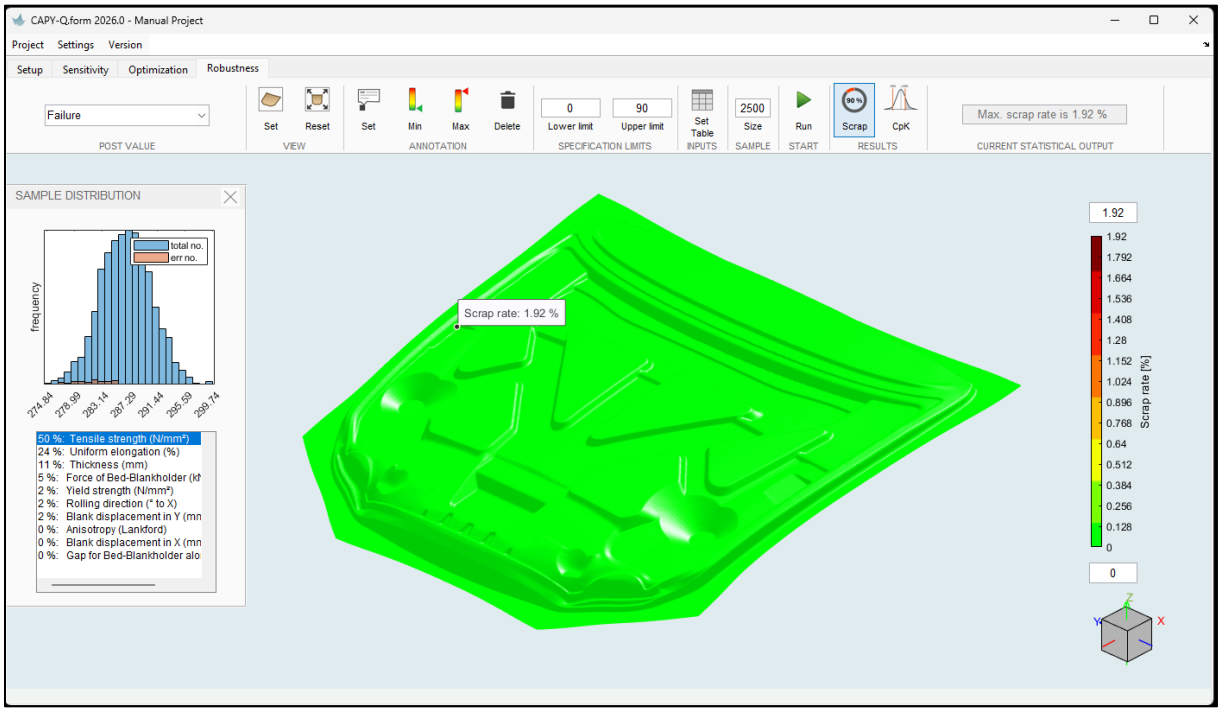
Workflow:

Step 1: The first step is to adjust the sliders manually to achieve the desired component quality in the **Parameter Optimization** panel. For optimization, it is recommended to use primarily the adjustable input parameters such as forces and distances/gaps. Depending on the loaded stage (e.g., OP10, spring back stage), both spring back or/and formability (failure, thinning, minor and major strain) can be analyzed.

The robustness parameters (strengths, uniform elongation, young modulus, thickness, friction) should be set according to the material supplier's specifications, but can also be varied if desired.

Step 2: In the second step, the adjustable parameters (forces, distances, blank displacement) are transferred to the robustness analysis via the "**Transfer**" button. These input parameter are kept constant at the values set in the sliders during the robustness analysis.

10.Module 4: Robustness Analysis

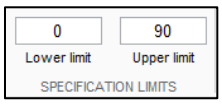


Functionality:

- Performing a robustness analysis for a selected simulation result in order to evaluate the capability of the forming process.
- Identifying the input parameters relevant to process robustness.

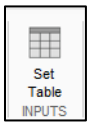
Key Components:

Specification Limits

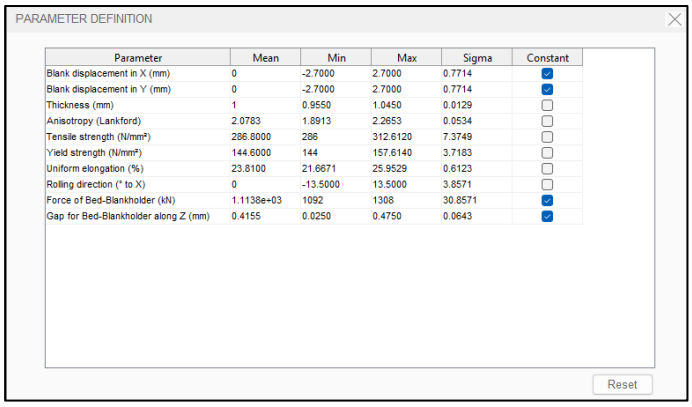


Define the **Specification Limits** (min and max) for the selected post value for the robustness analysis. Based on these limits, the number/distribution of defective parts is calculated (see Workflow - Step 1).

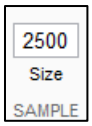
Set Table Button



The **Set Table Button** opens the table where the limits (Min/Max), expected value (Mean), and standard deviation (Sigma) of the input parameters are defined. Within these limits and standard deviation, the input parameters are scattered normally in the robustness analysis. Input parameters can also be kept constant by checking the "Constant" column. This is particularly useful for adjustable parameters such as forces and distances. The "Reset" button can be used to reset the table to its initial values (see Workflow - Step 2).



Sample Size Field

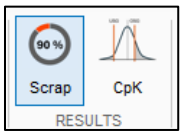


The **Sample Size Field** can be used to set the number of virtual simulation runs with which the robustness analysis is performed on the model (see Workflow - Step 3).

Run Button

The robustness analysis is started with the **Run button**

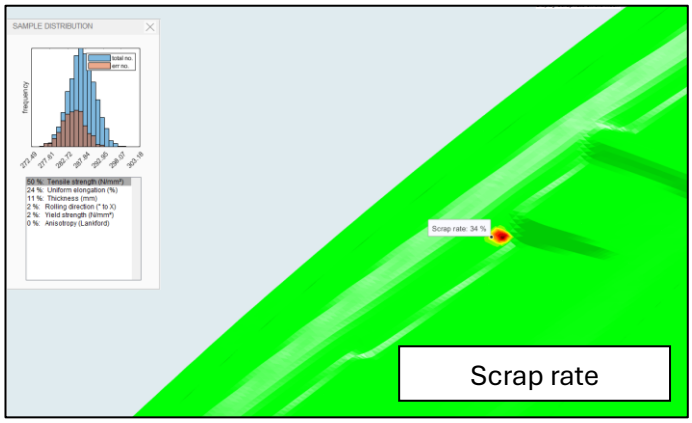
Results



Two different robustness results can be displayed in CAPY-Q. One is the **"Scrap"** rate and the other is the **"CpK"** value. Both results are calculated for each FE node of the formed part.

The **scrap rate** is a relative value between 0 % and 100 %. It indicates how many parts are outside the selected specification limit.

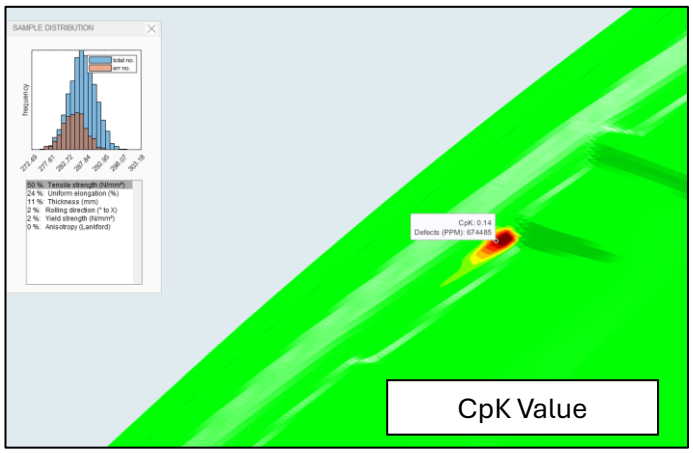
0 % scrap rate – all parts are within the specification limit.
100 % scrap rate – all parts are outside the limits.



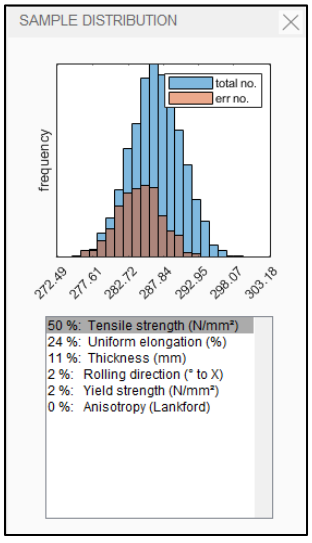
CpK (process capability index): This key figure evaluates how well a process complies with the specifications specified by the customer and how centered the process mean is between the tolerance limits. A higher value indicates a more capable process. A generally accepted minimum value for capable processes is Cpk ≥ 1.33 (see Workflow - Step 5).

The Cpk can also be used to calculate the failure rate per million formed parts produced. This value is called PPM (parts per million) and indicates the number of defective parts per million units produced. A lower PPM number means less scrap and better quality.

Cpk	PPM	Interpretation
0.67	approx. 1,350 ppm	Process not capable
1.00	approx. 270 ppm	Process at the limit of capability
1.33	approx. 32 ppm	Process capable
1.67	approx. 0.6 ppm	Process very capable
2.00	< 0.001 ppm	Process extremely capable



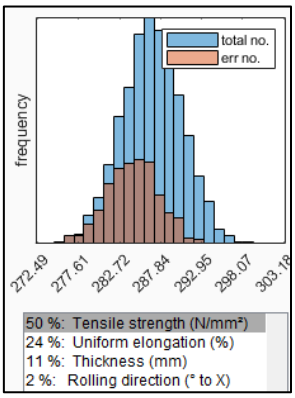
Sample Distribution



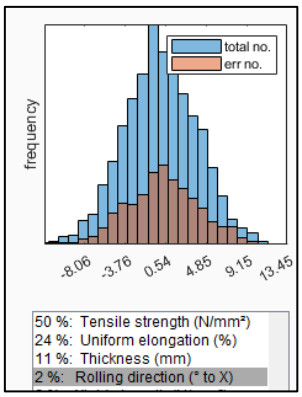
The "Sample Distribution" panel is always displayed when an annotation is set on the part surface in the Tab "Robustness analysis". It shows how the samples (robustness analysis runs) are distributed across the range of input parameters. The blue bars represent the total number of samples. The red bars indicate those samples that are outside the specification limits. If the scrap rate is 100 %, the red bars are exactly as high as the blue bars. If the scrap rate is 0 %, the red bars are not visible.

The various input parameters can be selected in the list box below. This also shows which input parameters are particularly important (high percentages are the important parameters) for the scrap rate and the CpK value.

Furthermore, you can see in which parameter limits the process is "robust" or "not robust." The figure shows the sample distribution for the input parameter "tensile strength", which has a significant effect on the forming result at 50 %. The failures (red bars) tend to be on the left (almost all failures between 272 and 290 N/mm^2). This means that with a tensile strength between 290 N/mm^2 and 298 N/mm^2, a robust process window is established and almost no failures are to be expected.



If an input parameter has no influence on process robustness, then the maxima of the blue and red bars on the X-axis are approximately the same value. This can be seen in the figure for the input parameter "Rolling direction" with only 2 % influence on the forming result and no shift of the red distribution curve to the left or right side.



Workflow:

Step 1: Define the **Specification Limits** (min and max) for the robustness analysis.

Step 2: Open the input table with the **Set Table Button** and check or define the limits (Min/Max), expected value (Mean), and standard deviation (Sigma) of the input parameters.

Step 3: Set the number of virtual simulations runs in the **Sample Size Field**.

Note: The default setting is to calculate 2500 examples. It is recommended not falling below this number, as a reduction weakens the statistical significance and reduces the calculation time only by a few seconds. An increase only changes the statistical significance in selected cases. In any case, we do not recommend calculating more than 10,000 examples, as the amount of result data generated may exceed the processing capacity of standard simulation PCs.

Step 4: Start the robustness analysis with the **Run button**.

Step 5: Result analysis (scrap rate and CpK).

11. Best Practices

- Always use consistent units (e.g., mm, kN, kN/mm²).
- Only activate parameters you intend to analyze to reduce computation time.
- Define realistic parameter limits to ensure meaningful simulation results.

12. CAPYcore algorithm

The CAPY-Q software uses methods and functions for modeling cause–effect relationships that fall under the definition of an "artificial intelligence system" as defined in Art. 3(1) of the EU Artificial Intelligence Act, because it applies mathematical techniques as described in Annex I of the Act.

The CAPYcore algorithm is a modeling method designed to analyze how multiple process inputs jointly influence a result. It automatically tests different mathematical model types—from simple linear equations to more flexible curve-fitting approaches—and selects the one that best fits the data. To prevent overfitting, part of the data is used for training and another part for testing.

The CAPYcore algorithm can represent complex behavior because it also uses weighted least squares, a technique that adapts the model locally to the data instead of relying on a single global formula. The method removes input variables with low influence to improve prediction quality. This reduction is based on sensitivity analysis, which measures how strongly each input affects the output. Variables with low influence are considered irrelevant and excluded.

The CAPYcore algorithm repeats the modeling process many times and averages the quality metrics to ensure stability. The model with the best overall prediction accuracy is then selected. The final model supports both understanding cause–effect relationships and optimizing process settings.

The model uncertainty indicated corresponds to the average error determined using cross-validation based on the training data. The actual uncertainty of individual model predictions may, in specific cases, be significantly higher. **All results are to be understood exclusively as supporting information and must not be interpreted as reliable, accurate, or guaranteed statements.**

13.Support & Contact

For technical questions or suggestions:

Email:

info@returnondata.de

Address:

Return on Data UG (haftungsbeschränkt)

Kirchstr. 16 e, D-01734 Rabenau