Multidisciplinary optimization for spot welding locations

Master's thesis in Product Development

SaiAkshay Tangella

Vignesh Devarajan

DEPARTMENT OF Industrial and Material science

CHALMERS UNIVERSITY OF TECHNOLOGY

Gothenburg, Sweden 2020

www.chalmers.se

Multidisciplinary optimization for spot welding locations

Master's in Product Development

SaiAkshay Tangella

Vignesh Devarajan

Department of Industrial and materials science
Division of product development
Chalmers University of Technology
Gothenburg, Sweden 2020

Multidisciplinary Optimization for Spot Welding locations

SAIAKSHAY TANGELLA

VIGNESH DEVARAJAN

EXAMINER: KRISTINA WÄRMEFJORD, Associate professor at the division of Product Development. Deputy Head of Department at the Department of Industrial and Materials Science

SUPERVISOR: ROHAM SADEGHI TABAR, Ph.D. Student at the division of Product Development

CHALMERS
UNIVERSITY OF TECHNOLOGY

Master's Thesis 2020
Department of Industrial and Materials Science
Chalmers University of Technology
Gothenburg, Sweden - 412 96, Telephone +46 31 772 100

Abstract

Automotive Body in white parts are joined using a method called spot welding. Previous studies have shown that the determined position of spot weld varies within ± 10mm from the nominal position [1]. The variation in the location of spot welds influences three disciplines, such as geometrical assurance, strength, and residual stress. This thesis aims to choose the position of spot weld with minimized geometrical variations and still to satisfy the requirements for strength and residual stresses. A multidisciplinary optimization method that incorporates all these disciplines to find the optimal position of a spot weld is developed.

This study was performed on an automotive test case model. This test case assembly consists of two parts, which are joined with seven spot welds. Individual modeling and simulation were performed on this assembly to study the effects of changing the positions of spot welds for all three disciplines. For this purpose, RD&T software was used to predict the geometrical variations, and Ansys workbench was used to analyze the strength and residual stresses for the test case model.

Matlab programming tool was used to develop a search algorithm that is used as an optimization algorithm during MDO and also to establish a connection between different software. An MDO workflow, which evaluates the different positions of spot weld to minimize geometrical variations and still satisfy the strength and residual stress requirements has been developed. From the MDO results, it is seen that the position of spot welds has an effect on all three disciplines, and therefore it is necessary to consider it during the development stage. The suggested optimal position of spot weld from this thesis has shown a 25% improvement compared to the nominal weld position.

Acknowledgement

Firstly, we would like to thank our supervisor Roham Sadeghi Tabar (Ph.D. student from the Industrial and Materials science department at Chalmers University of Technology), for his constant support and for being a great motivation for us during the challenging times.

Secondly, we would also like to thank our examiner, Kristina Wärmefjord (Associate professor from industrial and materials science department in Chalmers university of technology), for her valuable inputs and guidance that helped us throughout this thesis.

Lastly, we would like to thank our family members and friends for being a great support throughout this time.

Table of contents

List of figures

List of Tables

Lists of abbreviations

BIW – Body in White

MDO – Multidisciplinary Optimization

RD&T – Robust Design and Tolerancing

CAT – Computer Aided Tolerancing

FEA – Finite Element Analysis

CAD – Computer Aided Design

RSW – Resistance Spot Welding

ST – Solidification Temperature

SCC – Stress Corrosion Cracking

GA – Genetic Algorithm

IDF – Individual Discipline Feasible

MDF – Multi Discipline Feasible

RMS – Root Mean Square

RAM – Random Access Memory

1 Introduction

This chapter aims at providing insights into the background, along with the goals and objectives, followed by the Research questions to be answered, and delimitations of the thesis.

1.1 Project background

Geometrical assurance is a highly increasing topic within the automobile and aerospace industry [2]. Geometrical assurance is a method used to minimize the geometrical variations in assembled products. To ensure high quality, geometrical methods and tools are used early on in the product development phase [3]. The tools such as RD&T and CAT (Computer-Aided tolerancing) are used to evaluate variations and to ensure the robustness between different solutions.

The geometrical variations of assembled products are affected by many different factors, such as part variation, fixturing, the joining process, gravity, material properties, and collisions [3]. All these factors need to be considered during the analysis to minimize the difference between simulation results and results from physical tests.

This thesis focuses on developing an optimization method to minimize geometrical variations caused due to the joining process. Spot welding tool is one of the processes used to join Body in White (BIW) parts by different industries such as aerospace and automotive. During the joining process, the position, sequence, and the number of welds play an essential role in determining the geometrical outcome [3]. In general, the choice of position and sequence is based on requirements of the strength of the assembly and also the accessibility for the joining tools. The industries use different strategies to find the spot-welding locations to fulfil their requirements. A standard MDO method to find the optimal position of spot weld with minimized geometrical variation still satisfying the requirements of strength and residual stress is developed. Fig. 1, shows the disciplines that are considered for this thesis work.

This thesis work is done in collaboration with the research group 'Geometry Assurance and Robust design' at the Department of Industrial and Materials Science (IMS) at Chalmers, leading within geometrical assurance.

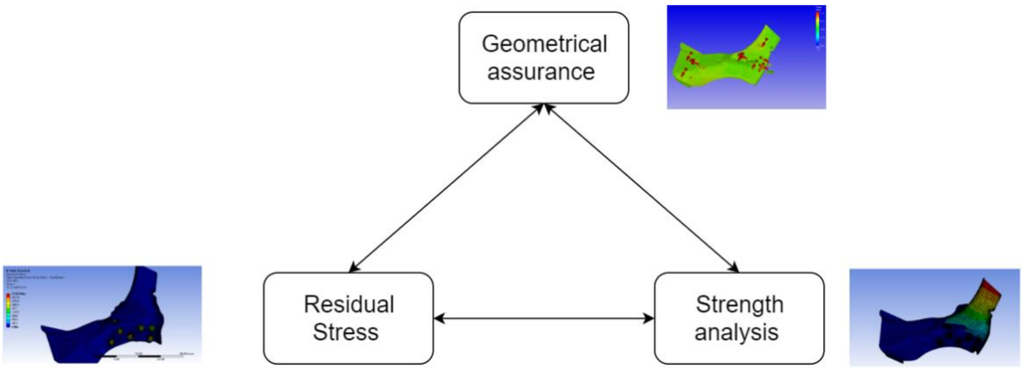


Fig. 1: Three different disciplines considered for optimization of spot welding location

1.2 Aim

Optimization of spot-welding locations with respect to geometrical variation, residual stress, and strength analysis using the MDO approach.

1.3 Goals

1. Finding out at least two suitable disciplines to find the best optimal spot weld position and combining with non-rigid variation simulation.
2. Developing an optimized formulation for spot weld locations by combining all disciplines such as geometrical assurance, strength analysis, and residual stress using a multidisciplinary optimization method with suitable algorithms.
3. Studying the optimized results and finding out the potential improvements in terms of accuracy and calculation efficiency.

1.4 Objectives

1. Literature review of non-rigid simulations (geometrical variations) of sheet metal BIW assembly using RD&T.
2. Literature review of each discipline (residual stress, strength analysis).
3. Literature review on multidisciplinary optimization (MDO) methods and algorithms (Genetic approach, conjugate gradient, weighted sum approach).
4. Data collection for geometrical quality of each discipline (position and sequence of weld points, positioning system).
5. Formulation of each discipline for simulating and optimizing the initial results.
6. Simulating and optimizing the results of each discipline by using different tools. RD&T tool for geometrical variation, FEA tools for structural analysis, thermal stress and optimizing the position of spot weld by using Matlab.
7. By using a multidisciplinary optimization (MDO) approach with suitable algorithms to optimize the location of spot weld with respect to geometrical variance, residual stress and strength analysis.

1.5 Research questions

Based on the defined goals and objectives, the following research questions were formulated.

1. How strength analysis and residual stress discipline, influence the optimization of spot weld location with respect to geometrical variation and their differences in the results of an individual and multidisciplinary optimization?

2. How to incorporate the findings or method developed from this thesis for different geometrical assemblies in order to find the optimized position of spot weld?

3. How selection of software platforms for each discipline plays an important role to perform multidisciplinary optimization?

1.6 Delimitations

The delimitations of this thesis are as follows

1. The sequence and the number of spot welds are fixed as it will increase the overall simulation time during the MDO process.

2. The framework for the multidisciplinary method will only be tested on an assembly given by the organization.

3. To perform MDO, the selection of software for individual discipline is limited which is discussed in Chapter 8.

2 Literature review

This chapter provides the reader with the insights of the literature relevant to carry out the studies for this thesis topic.

2.1 Sheet Metal Assembly

Sheet metal assemblies are used in a wide range of products from automotive, aerospace, medical, and home appliances. Since this thesis study deals with the optimization of spot weld locations for sheet metal assemblies, this section gives a comprehensive insight into the sheet metal assembly process. Generally, the sheet metal assembly process is mainly divided into two stages as shown in Fig. 2.

1. Forming individual sheet metal components
2. Assembly of all individual sheet metal components in a final assembled product structure.

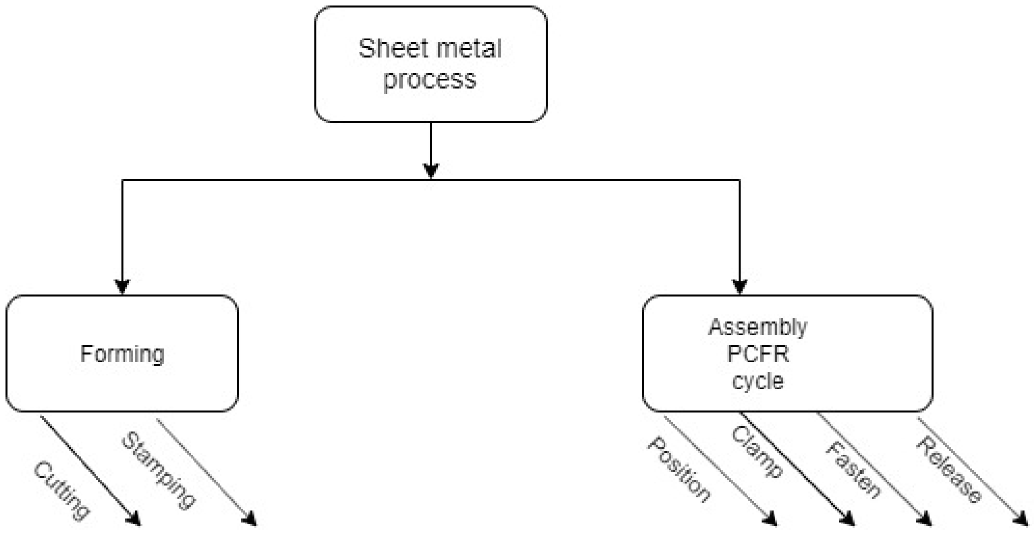


Fig. 2: Sheet metal process

Further, the sheet metal forming process is divided into multiple stages of stamping, which includes cutting and bending, and finally, the assembly process, which is divided into a four-step cycle called the PCFR cycle: Place, Clamp, Fixture, and Release [4] [5].

In the PCFR cycle, the sheet metal components are positioned in the assembly fixtures, which are secured by the locators. The fixtures are mated into locators; then, the individual parts are clamped together. Deformations occur in this step. The next step is to fasten, which is done using spot welding where the weld gun closes the remaining gap and joins the individual sheet metal components. There can be deformation seen in this step. Finally, the clamps and fixtures are released (Spring back is allowed), and the assembled structure is allowed to deform until the unbalanced stress state caused during the assembly process is reached to the balanced stress state, But still, the residual stress is inherent inside the structure which may affect fatigue life under loading conditions [4].

2.1.1 BIW Components

As mentioned in the previous section, the Sheet metal assembly is used in different industries, but the thesis focuses mainly on automotive BIW components. Therefore, this section is introduced to have a general understanding and classification of BIW components.

BIW or Body in White components refers to the stage in automotive design and manufacturing where the sheet metal body structure including closures (All door panels, hoods, and bonnet), roof structures(roof panels, bows, and headers) are assembled but before engine, wheels, interior trims (Dashboard, seat), exterior trims (plastic trims parts, roof liners, door handles) and electronics (wiring harness, entertainment system) are added to the body frame structure, as shown in Fig. 3. During the manufacturing phase, the sheet metal is allowed to undergo several different manufacturing operations to finally become a unique body part which is designed and integrated. This process is done before the painting job, and due to its metallic appearance of the body structure, it is known as the body in white [6].

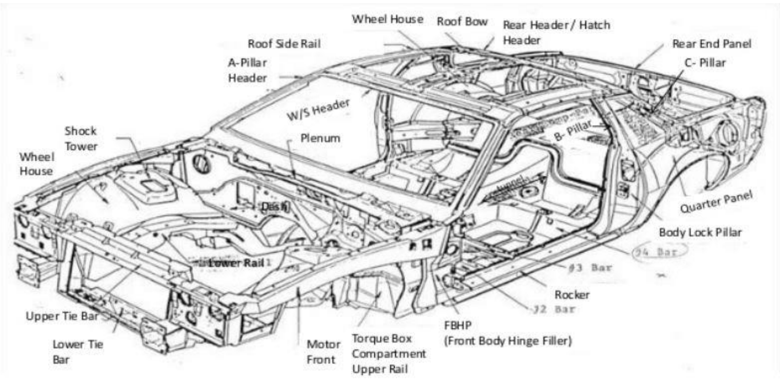


Fig. 3: BIW (Body in White) parts [6]

Body in White or BIW is further classified into the following sub-structures.

1. **Under Body Structure**:(Front and rear floor panels, dash partition panels, rear wheelhouse)
2. **Side body structure:** (‘A’ pillar, ‘B’ pillar, ‘C’ Pillar, Quarter Panel, Ring panel)
3. **Roof and closures:**(door panels, hood, roof panels, bonnet)
4. **Front and rear end structures:** (fenders, wing panels, tie bars, cowl, plenum panels)

Reinforcement components: These are the components that are used to provide strength and stiffness to other bodies in white components. The test case model used in this thesis study is a reinforcement component.

2.2 Joining process

2.2.1 Resistance Spot welding

Resistance spot welding is one of the critical joining processes in an automotive industry which is widely used for joining sheet metal assemblies. In general, an automotive body assembly requires around 4000-5000 spot welds depending upon the size [7]. Due to its high welding efficiency and suitability for automation, it is considered to be the most effective and commonly used joining process. In this process, two or more sheet metals of a thickness up to 3mm are held together by applying the welding pressure, as shown in Fig. 4. A contact is created through which an electric current is passed through the electrodes. In the first cycle, the heat generated due to electrical resistance that will melt the metal and fuses together. In the second cycle, the electric current is turned off, but the pressure is still maintained until the molten metal solidifies to form a joint in between the sheet metal [8].

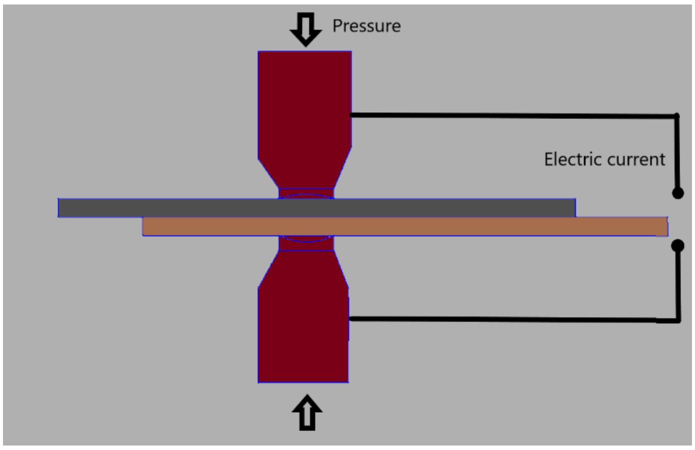


Fig. 4: Schematic diagram of resistance spot welding

Resistance spot welding works on the principle, which states that the amount of generated is directly proportional to the electrical resistance, the thermal conductivity of the material, and the amount of time current is applied.

$$Q = i^2RT \tag{1}$$

Q= heat generated (joules); i = current applied (Amperes); R = electrical resistance of the material (Ohms); T = Time duration of current (sec)

Copper electrodes are used because of their high thermal conductivity and low electric resistance, which allows generating more heat in the workpiece rather in the electrode.

2.2.2 Stages of Resistance spot welding (RSW)

The resistance spot welding consists of the following stages [9] as shown in Fig. 5.

1. **Squeeze Time**

   This is the stage where the electrodes are brought in contact with the sheet metal, and the welding force is applied to the contact surface to close the gap between the sheet metal. Here, in this stage welding force and squeeze time are controlled parameters.

2. **Welding time**

   This is the stage where the electric current is applied at the contact region through electrodes and the resistance is created by the material at the interface, which is responsible for heat generation to form a molten nugget. Here, in this stage, welding time and current are controlled parameters.

3. **Hold time**

   In this stage, the current supply is turned off and the sheet metal is under constant electrode pressure to allow the molten nugget to cool down and solidify to a fully-grown nugget. Here, in this stage, the electrode pressure and hold time are controlled parameters.

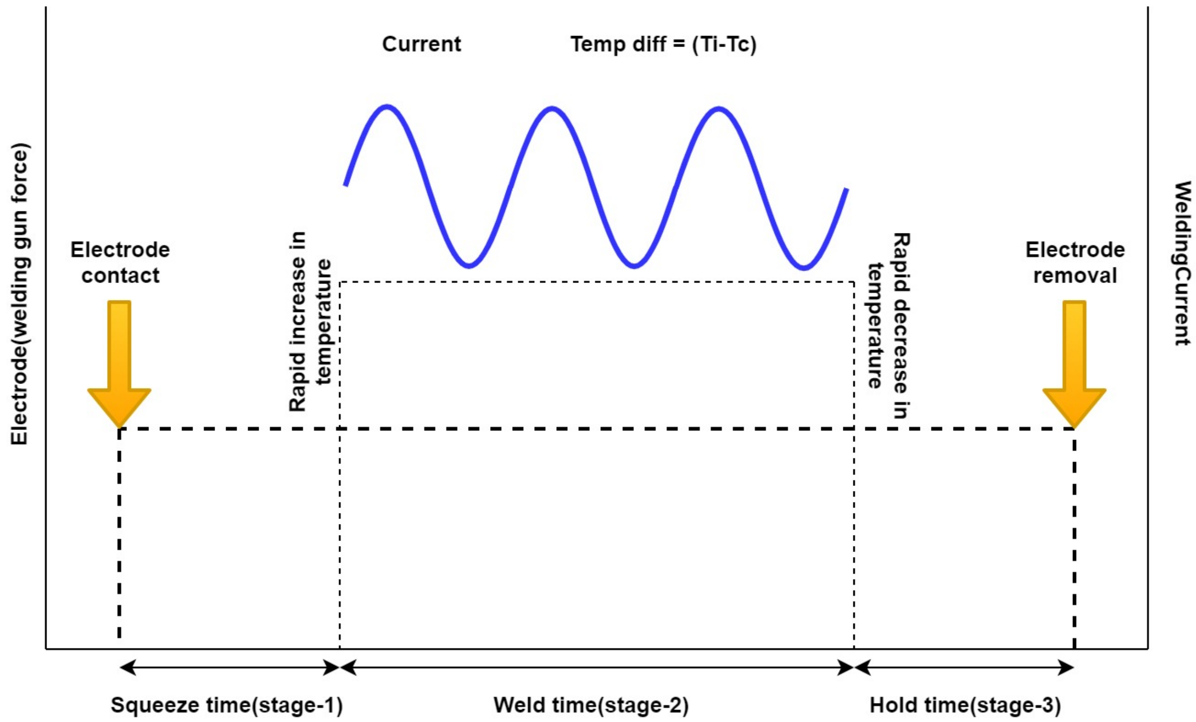


Fig. 5: Graphical representation of joining Process [9].

2.2.3 Parameters that influence the geometrical outcome

The parameters that influence the geometrical outcome of the final product are,

1. Position

During the joining process, two sheet metals are connected by applying equal forces from each side of the sheet metals through balanced welding gun. Therefore, allowing the sheet metals to meet at a position of equilibrium to spot weld [1].

The position of spot weld has a significant influence on the geometrical variation of final assembly and affects the characteristics of the final assembly as shown in Fig. 6. The following reasons cause the spot weld position variation.

- The incoming part variation in the areas where spot welds are located.
- Wear and tear of electrodes.
- Variation in locating schemes of the parts to be assembled.
- Lack of repeatability of the robot during the welding process.

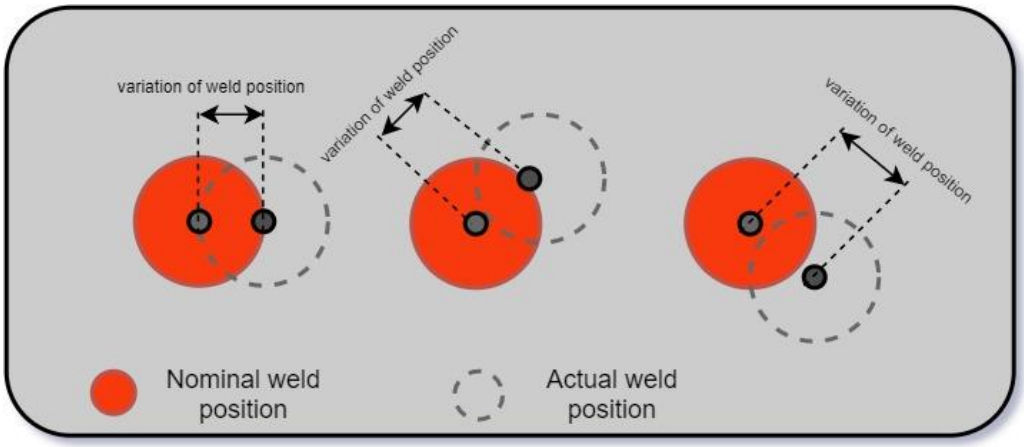


Fig. 6: The actual spot weld position vs. nominal spot weld position

2. Sequence

The sequence is the order in which the spot welds are welded in the sheet metal assembly. It has a significant effect on the geometrical variation of the final assembly and also has an impact on residual stresses during the joining process. In order to minimize the variation in the final product, an optimal sequence is to be selected [7].

3.Number

The number of spot welds has a significant impact on the geometrical variation of the final assembly and also has an effect on performance characteristics, such as dynamic, static and crash behaviour. An optimal number of spots welds needs to be selected without affecting the rigidity of the structure. Also, an increase in the number of spot weld has direct impact on the cost of production [8].

2.3 Geometrical variation

Product design is said to be robust when the variation in the product is minimized without eliminating the causes of variation [11]. To achieve a robust product design, the performance level should be maintained even if there is any significant part variation seen in the incoming parts, due to variation seen in the manufacturing process these incoming part variation may cause misalignments and deformation during fixturing which may lead to assembly variation [3]. Assembly variation is also caused due to fixturing (locating schemes) and the joining process (weld distortions), as shown in Fig. 7.

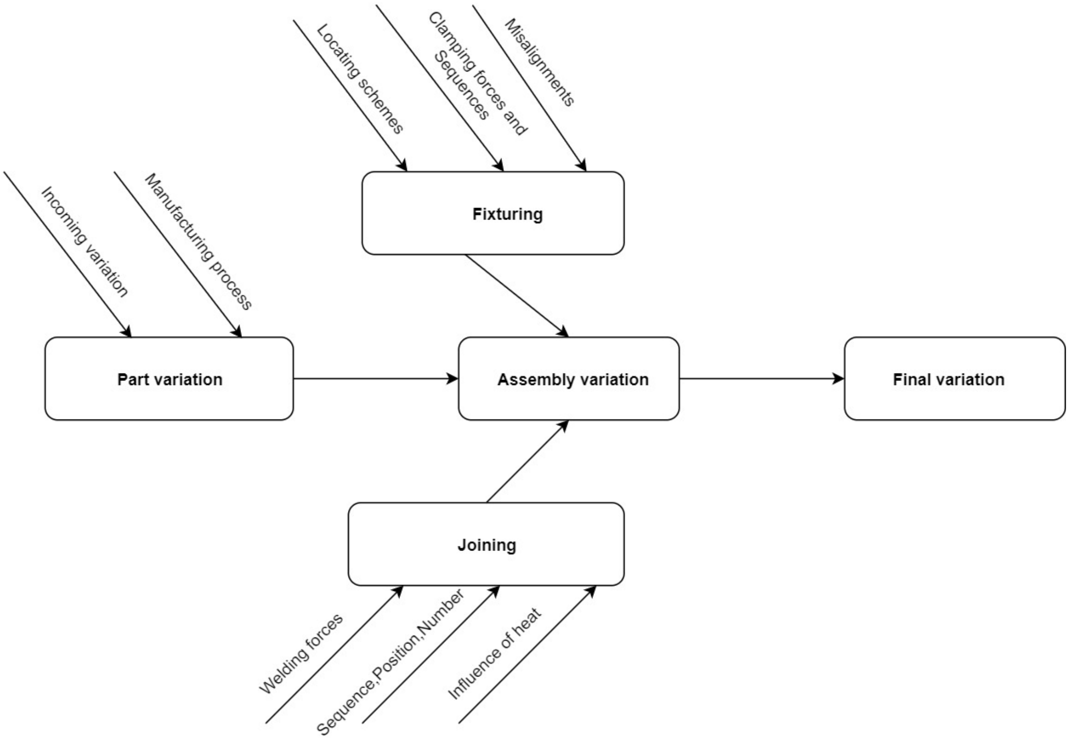


Fig. 7: Factors affecting geometrical variation

2.4 Non-rigid variation simulation

Variation simulation is performed during the product development stage to reduce lead time and to increase the robustness of the final subassembly [3]. Non-rigid variation simulation is preferred over rigid to improve the accuracy and also the ability to over constrain parts and assemblies.

Steps involved in including joining sequence for non-rigid variation simulation are [12]

1. The sheet metal parts are positioned using the locating schemes (N-2-1)
2. Clamping the parts to the nominal position
3. Weld points to be defined.
4. Joining variables (position and sequence).
5. Contact points to prevent the parts from penetrating through each other.
6. Assembly spring back action

Different kinds of joining methods such as clip fasteners, spot weld, riveting are used in joining the sheet metal parts. Spot weld is the most commonly used joining method in the automotive industry. The processes involved in the joining of the sheet metal parts, as shown in Fig. 8, are first, the parts are loaded on the holding fixture where the part is locked in all six degrees of freedom. Secondly, the parts are clamped to the nominal position. The assembly robot joins the parts using spot welder on the specified location. Lastly, the assembly is allowed to spring back from the holding fixture [12].

To predict assembly variation, sources of part variation is provided as input for the simulation. The methods such as Worst case, Root mean square, and Direct Monte Carlo simulation are the different techniques used to predict the variation in the final assembly. The direct Monte Carlo simulation, along with FEA, is a standard technique for performing variation simulation. This method has few disadvantages, such as it requires a more significant number of runs, computationally expensive and time-consuming. The method of influence coefficient developed by Liu and Hu overcomes all the disadvantages. The unique feature of this method is that it generates a sensitivity matrix, which is a linear relationship between part deviation and the assembly spring back deviations [12].

$$U = (S)(V) \tag{2}$$

Where u is Spring back from the assembly fixture, S is the sensitivity matrix, and V is the source of part deviation. The linear relationship derived from the FEM along with Monte Carlo simulation can be performed, requiring a lesser number of runs and time. This method predicts not only the assembly deviation but also the percentage of rejects of assembly based on the distribution of parts.

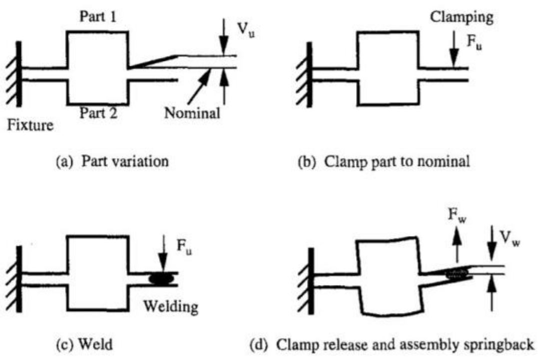


Fig. 8: Steps to be considered during non-rigid variation simulation [12]

2.5 Stress Analysis

Residual Stress is the inherent stress which is present in the weld joint without application of external loading. These stresses occur during the development of the weld joint. The differential volumetric change (or) differential expansion and contraction of the metal internally affect the material properties, which lead to the residual stresses [13].

In Resistance spot welding, heat is generated by applying force and current on the sheet metal, which allows the metal to melt and form a molten nugget, which cools down under constant pressure to form a joint. During this process, the heat is dissipated to the surrounding region of the metal as shown in Fig. 9, [8] [13].

The different zones around the welds are subjected to different values of a rise in temperature as shown in Fig. 10. So generally, this region should expand on heating and contract (shrink) during the cooling cycle thus experiencing local strains (change in length) in the different regions. The volumetric change is restricted as these zones are an integral part of the sheet metal and clamping of the sheet metal. Since the volumetric changes are restricted that is Strain, residual stresses are developed in the sheet metal. The maximum restriction is seen in the region closer to the weld area, causing more residual stresses [13].

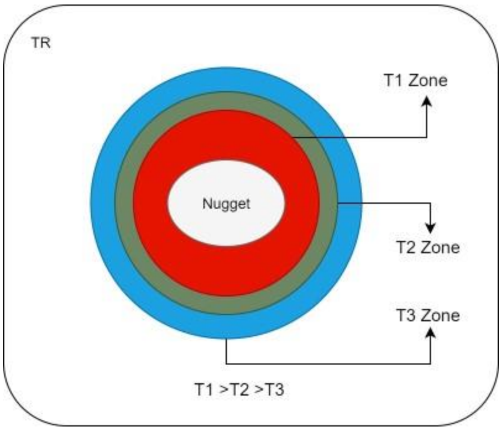


Fig. 9: Strain at different temperature zones

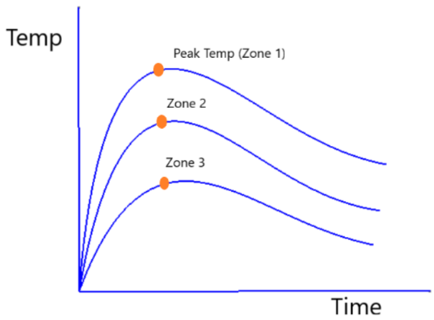


Fig. 10: Peak temperature graph

It is seen that different zones will be expanding to different magnitudes due to differential heating. This difference is causing to form a compressive strain. While cooling, different zones shrink at different magnitudes causing tensile strains which ultimately lead to the compressive stresses. Finally, there are some tensile stresses left in the metal which may affect the mechanical properties of the metal [8] [13].

$$\text{Residual stresses} = \text{elastic strain } (\epsilon) * \text{modulus of elasticity } (E) \quad (3)$$

The maximum magnitude of the residual stresses depends upon the maximum elastic stress. If it more than the maximum value, it will result in plastic deformation.

Factors affecting residual stresses are

1. Material properties ($\boldsymbol{\alpha}$) = thermal expansion coefficient
2. solidification temperature (ST)
3. weld joint (nugget size)
4. welding sequence
5. Clamping and fixturing

Residual stresses affecting mechanical properties of metal:

1. The metal is prone to brittle nature due to residual stresses at low temperatures.
2. Residual stress increases the stress corrosion cracking (SCC).
3. Tensile residual stress decreases the fatigue strength of the metal.

2.6 Multidisciplinary design optimization

Most of the modern engineering systems are multidisciplinary, and the analysis is often very complex, involving hundreds of computational simulations. Also, members from different cross-functional teams, different parts of the world which makes it very difficult for most companies to manage the design phase of the product development and make critical decisions. This is where the MDO systems come in to play to find the optimal solution of the whole system and still satisfying the individual discipline and time constraints. This section provides the reader clear understanding of MDO, how two or more disciplines analysis are coupled together with optimization problems and also different MDO formulations.

2.6.1 General Problem Formulation

The objective function, design variables, and constraints form the optimization problem statement:

$$\begin{aligned} &\text{Minimize} && f\,(x, p) \\ &\text{With respect to} && x \in R \,\hat{}\, n \\ &\text{Subject to} && h\,(x, p) = 0, j = 1, 2, \ldots, m \\ & && g(x, p) \geq 0, k = 1, 2, \ldots, m \end{aligned} \tag{3}$$

f: objective function, output.

x: design variables vector.

h: equality constraints vector.

g: inequality constraints vector.

2.6.2 Objective Function

An objective function is used to compare two designs based on what do we want to minimize. If we select the wrong goal, it does not matter how functional the analysis is, or how efficient the optimization method is. Therefore, it is essential to select a relative objective function [14]. The objective function may be linear or nonlinear and may or not be given explicitly. We will represent the objective function by the scalar f (objective function) [14] [15].

2.6.3 Design Variables

Design variables are also known as design parameters and are represented by the vector x [14]. They are the variables in the problem that we allow for varying in the design process. Optimization is the process of choosing the design variables that yield an optimum design. Design variables should be independent of each other. Design variables can be continuous or discrete. Discrete variables are sometimes integer variables.

2.6.4 Constraints

As the objective function, constraints can be linear or nonlinear and may or may not be given in an explicit form. They may be equality or inequality constraints. Constraints on the design variables are called bounds and are easy to enforce. At a given design point, constraints may be active or inactive. This distinction is particularly important at the optimum [14] [15].

2.7 Algorithm

Genetic Algorithms (GA) are adaptive heuristic search method developed in the early 1970's which mimics the process of natural biological evolution applying the principle of survival of the fittest, where the fittest individuals (potential possible solution) are reproduced (crossover and mutation) to a new generation (fitter optimal solution) [16] [17], see Fig. 11. These algorithms are used to generate high-quality solutions for optimization problems.

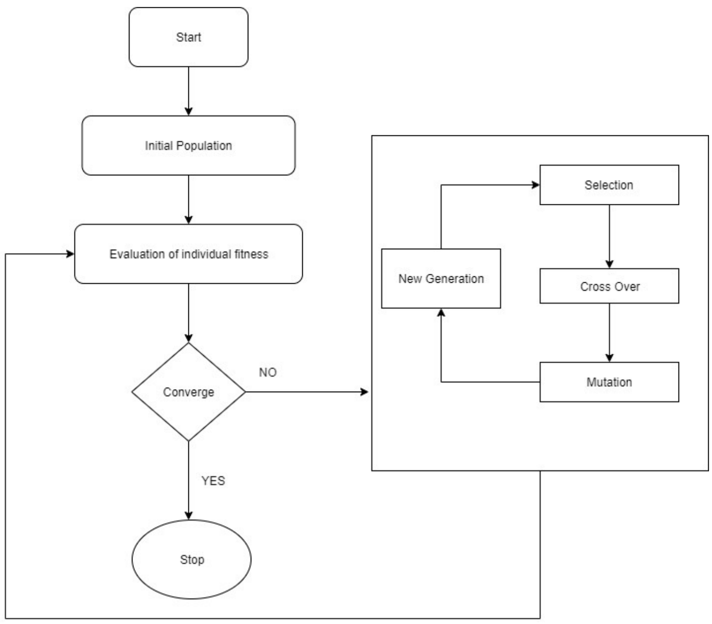


Fig. 11: Genetic algorithm

The operators of the genetic algorithm:

Selection: The idea behind the selection phase is to select the potential possible individual's (Parents) based on their fitness function Score.

Crossover: The idea behind the crossover phase is to create a new offspring by the transfer of genetic material from two parents. These new generation offspring possess better characteristics features than that of parents.

Mutation: The idea behind the mutation phase is to create a genetic diversity by altering one or more genes from the generated offspring.

2.8 Multi-Disciplinary Optimization(MDO) Formulation

MDO formulation is generally used to specify the objective, the constraints, and the design and coupling variables for an optimization problem. All the formulations specified in this thesis yield to non-linear programming problems, but different formulations have different attributes to solve an MDO problem. An Algorithm is generally a sequence of steps that are carried out to solve an MDO problem. The different types of MDO formulations are

2.8.1 Multi-discipline feasible

Multi-discipline feasible (MDF) is the traditional and most common way used to formulate the MDO problems. The optimizer is used to provide a design variable vector (X) to the combined coupled system (all disciplines) to perform a complete multidisciplinary feasible analysis as shown in Fig. 12, [15].

MDF formulation is simple and basic as the coupled relationship is solved inside the system, and the sate variables, consistency, and analysis constraints are eliminated in the optimized problem. Single disciplinary optimization techniques can be used as the different disciplines are coupled into a single multidisciplinary analysis [14] [15].

MDF FORMULATION:

Minimize $f[\mathrm{X}, \mathrm{Y}_1(\mathrm{X}), \mathrm{Y}_2(X)]$ (4)

With respect to ‘X’

Subject to $\mathrm{C}\big(\mathrm{X}, \mathrm{Y}_1(\mathrm{X}), \mathrm{Y}_2(\mathrm{X})\big) \geq 0$

$\mathrm{C}_1\big(\mathrm{X}, \mathrm{Y}_1(\mathrm{X}), \mathrm{Y}_2(\mathrm{X})\big) \geq 0$

X: Problem design variable inputs. $\mathrm{Y}_1, \mathrm{Y}_2$: Analysis output.
C: Design Constraints (Global) C_1: Local constraints.

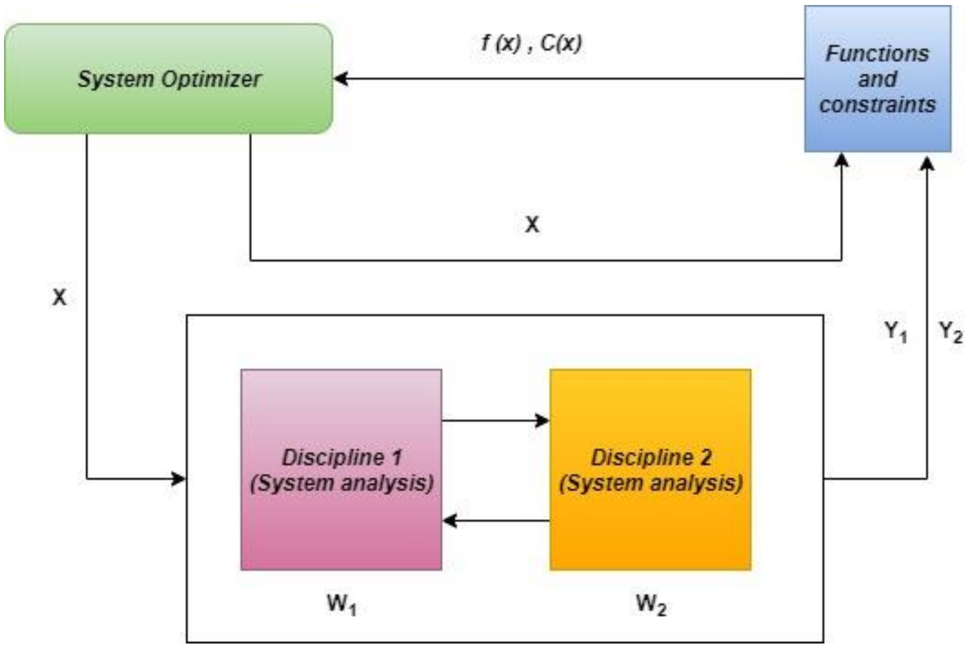


Fig. 12: Multidiscipline feasible [15]

Advantages:

- MDF Reduces the problem size when the number of coupling variables is high, which may even overshadow the lower degree of parallelism.

Limitations:

- Computational cost is significantly high is due to, lack of parallel computation.
- The overall speed of the optimization is significantly slow.

2.8.2 Individual discipline feasible (IDF)

Individual discipline feasible (IDF) is the decoupled part of Multidiscipline feasible (MDF), where the system optimizer is used to perform the interdisciplinary consistency by driving the individual disciplines towards multidisciplinary feasibility [14].

IDF performs individual discipline feasibility at each design point instead of performing multidisciplinary feasibility, which allows the solution of each discipline to be feasible, but the whole system may not be feasible. So, a solution for the entire system is only feasible at the end [14] [18]. Each discipline will have a feasible design even after the optimization stops prematurely but for the whole system, the coupling variables(Xij) may not have converged resulting in an infeasible design.

To decouple the disciplines, coupling variables are introduced to the design variables so the disciplines do not rely on each other, as shown in Fig. 13. Some equality constraints are introduced for each coupling variable to make sure that the system feasible is reached to an optimum solution [15].

IDF Formulation:

$$\text{Minimize} \quad f[X, Y_1(X), Y_2(X)] \tag{5}$$

With respect to 'X'

$$C\big(X,\ Y_1(X), Y_2(X)\big) \geq 0$$

$$C_{12} \equiv X_{12} - Z_{12}\,(Y_2(X, X_{21}) = 0$$

$$C_{21} \equiv X_{21} - Z_{21}(Y_1(X, X_{12}\,) = 0$$

X: Problem design variable inputs.

Y_1, Y_2: Analysis output.

C : Design Contraints.

W_1 , W_2 : Governing equations.

X_{12} , X_{21}: Coupling design variables inputs.

Z_{12}, Z_{21} : Coupling variables output.

C_{12}, C_{21}: Coupling constraints.

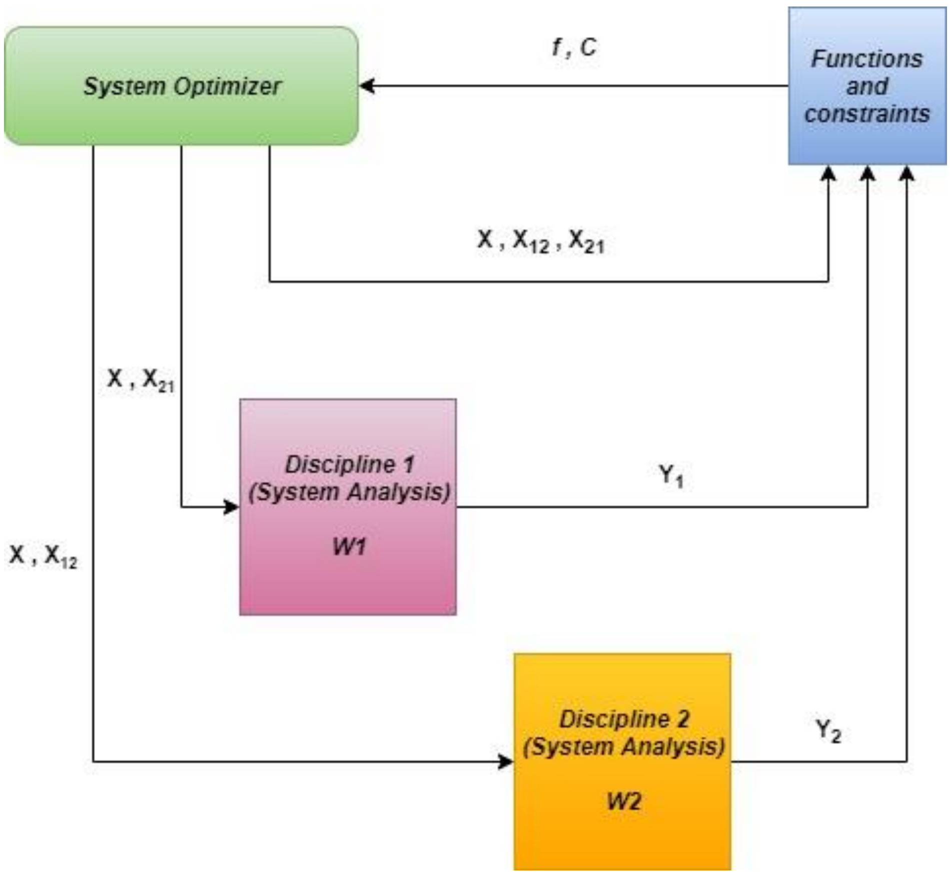


Fig. 13: Individual discipline feasible [15].

Advantages:

- When compared to other methods involved in multidisciplinary analysis, IDF is the most robust method
- As the governing equations are decoupled, they can be solved in parallel, increasing the overall performance of the optimization.

Limitations:

- The cost of sensitive analysis per design point can be significant, as the number of design variables and constraints is increased.

2.8.3 The comparison between MDF and IDF

Attributes	***IDF***	***MDF***
Discipline feasibility	Feasibility of individual discipline at each design point	Feasibility of multidiscipline at each design point
Computational time	Less	High
Optimization problem formulation size	Medium	Small
Optimization speed	Fast	Slow
Robustness	High	Low

Table 1: Comparison between IDF and MDF

3 Simulation Methodology

3.1 Introduction

Over time the computer simulation has shown greater interest among different industries such as medical, aerospace, and automotive to support the development process. The model is created based on mathematical expression or algorithm, often referred to as the approximation model of a real-world problem. John A Sokolowski [19] relates to simulation as a method of extracting information about the behaviour of the model when it is executed. To perform a simulation, several steps are involved in understanding and solving real-world problems. A simulation model not only solves the problem but also allows the user to view the result in 2D or 3D visually. The advantages of performing simulation are saved cost and time, result visualization, and handle uncertainty during the development process. The next section will focus on the different steps involved in a simulation study.

3.2 Different steps involved in a simulation study

The different steps involved in a simulation study based on Banks et al. (1996) are followed in this thesis [20]. Fig. 14, and Fig 15, show the steps involved in a simulation study.

1. Problem formulation

The first step involved in a simulation study is to study and understand the problem. The concepts behind the problem need to be well understood even before the formulation. In this thesis, the positions of the spot weld affect three disciplines, such as geometrical variation, strength analysis, and residual stresses. A literature study was conducted to study and understand the problem.

2. The setting of objectives and project plan

Once the problem is well understood, the immediate next step involves setting the objectives, goals, and timeline for this thesis. The objectives shall include the research questions that are needed to be answered through this study. The timeframe of the study shall consist of the various milestones and deliverables that are required for a simulation study.

3. Model conceptualization

In this stage, the complexity of a real-world simulation problem can be replaced with a conceptual model. This will enhance the quality of the final simulation result. The same strategy is followed in this thesis where the simulation study was initially conducted in a conceptual model before dealing with a complex model that will be discussed in the coming chapters.

4. Data collection

Data collection is an essential stage that will play a significant role in both the timeline and final quality of the simulation study. In this thesis, the essential data regarding the standards of spot weld, and the process involved in the joining process of sheet metal assemblies are obtained through a literature study. Also, the types of software that can be used for each discipline are studied.

5. Model translation

The conceptual model created in step 3 is translated into an operational model. The reference model that is used in the simulation study is provided to different software platforms to analyze such as Geometrical variation in RD&T and Residual stress and Strength analysis in Ansys Workbench 2019

6. Verification and validation

The verification of a conceptual model is analyzed in this step. Once the model is built on its respective software platform, it is then validated before the individual simulation. The simulation is performed based on part variation for geometry assurance, Von mises stress for residual stress and total deformation for Strength analysis. The results from each individual simulation based on the Nominal weld position is collected.

7. Analysis and experimentation

The insights from this individual simulation help formulate the Multidisciplinary optimization problem. The simulation method to perform multidisciplinary optimization including each discipline is developed. The findings and conclusions of Individual simulation are the initial input data required to model the MDO problem. The MDO problem is built on MATLAB R2018b software platform, which establishes an interface between RD&T (Geometrical variation) and Ansys workbench (Residual stress and Strength analysis). The model is then verified for any potential errors in RAM allocation or Disk space and also to cross verify whether the exchange of data between each discipline is working as it is planned. Initially, the simulation is performed for a minimum number of iterations, i.e., ten each time with a new set of weld points to validate the result. This helped identify any potential error in formulation before simulating the max number of iterations which in turn saved a lot of time. After the simulation is performed for the max number of iterations, the result i.e., the optimized position of a spot weld is once again subjected to individual simulation with a newly updated position and their obtained respective disciplines values are verified against it.

8. Report and Documentation

The documentation of the whole simulation study is essential for future purposes. The important finding and simulation techniques, if reported clearly, will save a lot of time for the reader who wishes to continue this study. Hence the whole simulation process involved in this study is provided as a guide for this thesis. Also, a report that explains the methodology and results from the optimization is presented.

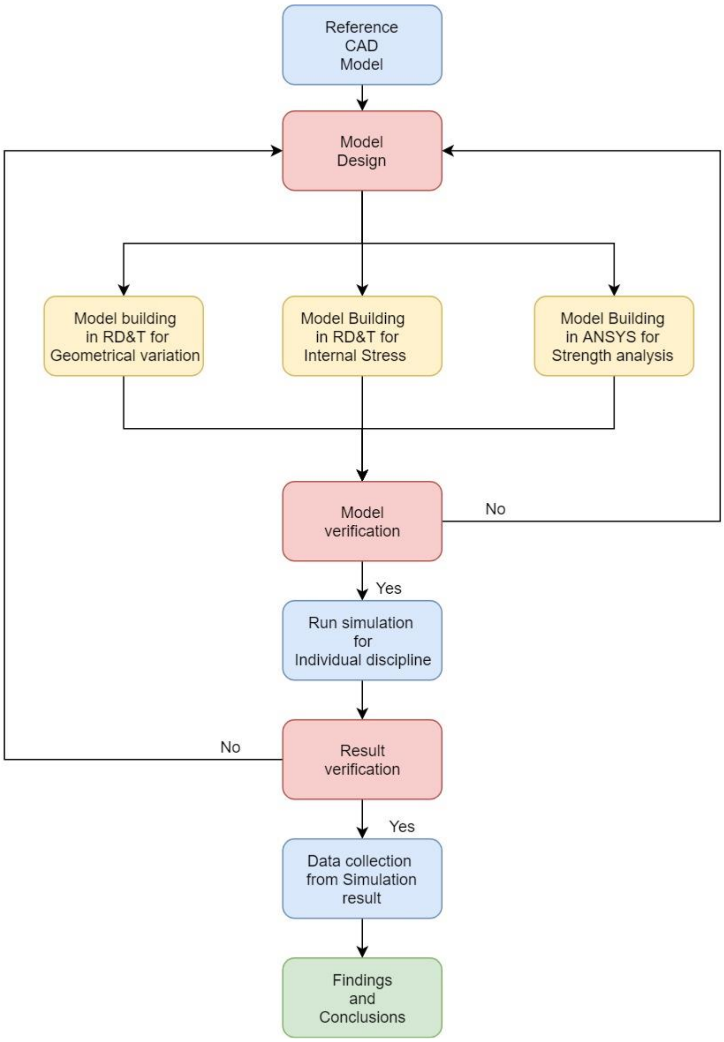


Fig. 14: Simulation method used for individual discipline

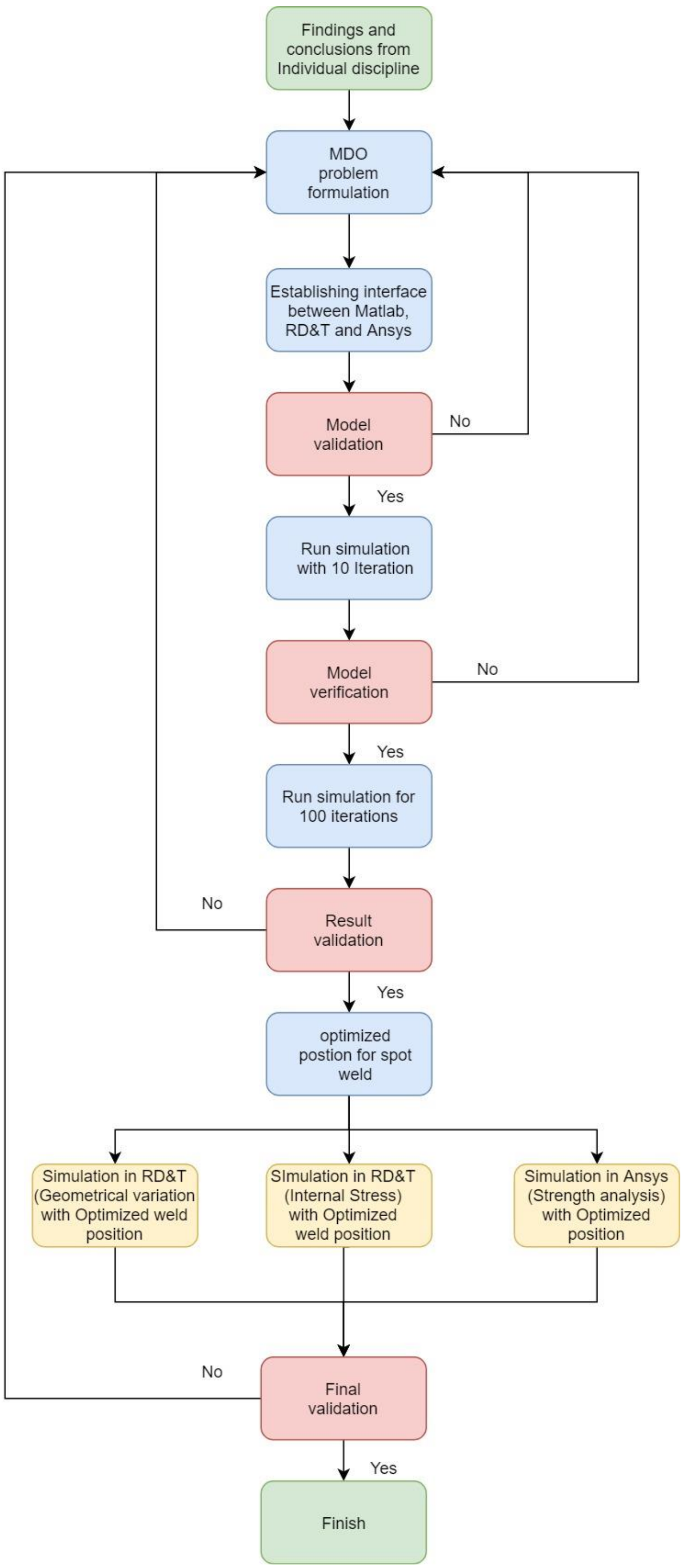


Fig. 15: Simulation method for Multidisciplinary Design Optimization

4 Reference CAD Models

4.1 Introduction

CAD model used for optimizing the position of the spot weld is a reinforcement part. This model is referred as the test case throughout this report. The test case model in total consists of seven-spot weld points to join the components. To start with individual discipline, the number of weld points is quite large for the test case model to make changes in the position of spot weld manually each time in every discipline. Therefore, a simple model with a single weld point is created to understand the theory behind the changes in each discipline with respect to the position of spot weld points. The CAD models created for this thesis to test are simplified versions of a reinforcement part for a Body in white component.

4.2 CAD Models

The initial three reference models such as L shape, U shape, C Shape is designed to have different numbers of weld points and to have weld points on various surfaces such as Flat and Curved. Therefore, time is saved in understanding the changes in Geometrical variation, Internal Stress, Strength of a subassembly with respect to the position of the spot weld.

1.L shaped model which consists of two flat sheet metal is formed to 90 degrees and welded through a spot weld method as shown in Fig. 16.

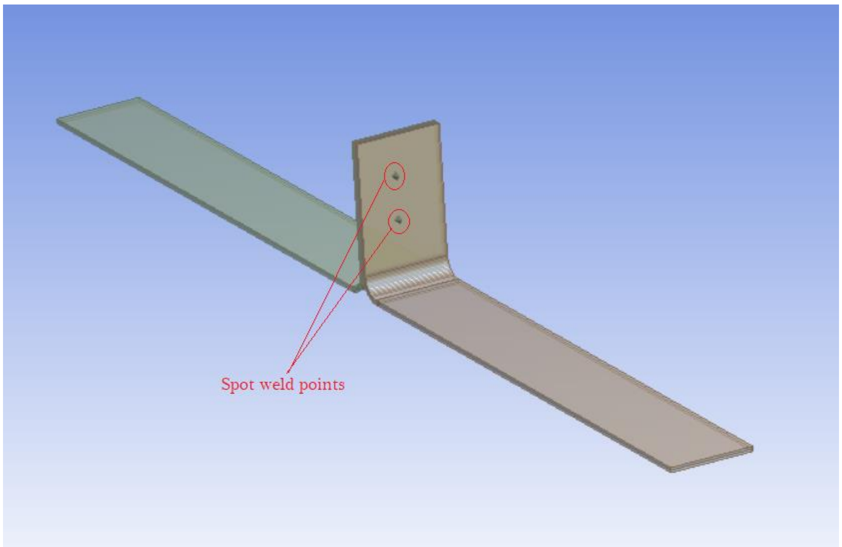


Fig. 16: L shape model

2. U shape model, which consists of two sheet metal, undergoes a forming to form a U shape, and later, the parts are joined through eight spot weld points, as shown in Fig. 17.

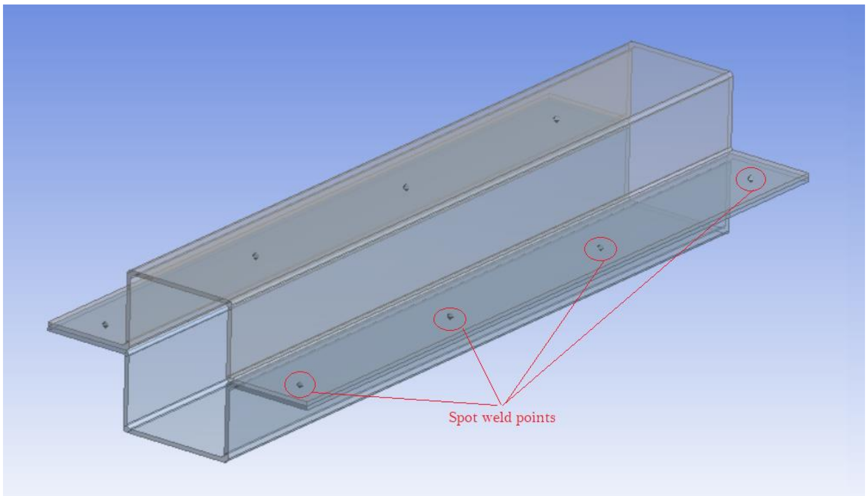


Fig. 17: U shape model

3. C shape, which consists of three weld points on a curved surface. The reference model is created using Catia V5 software, as shown in Fig. 18.

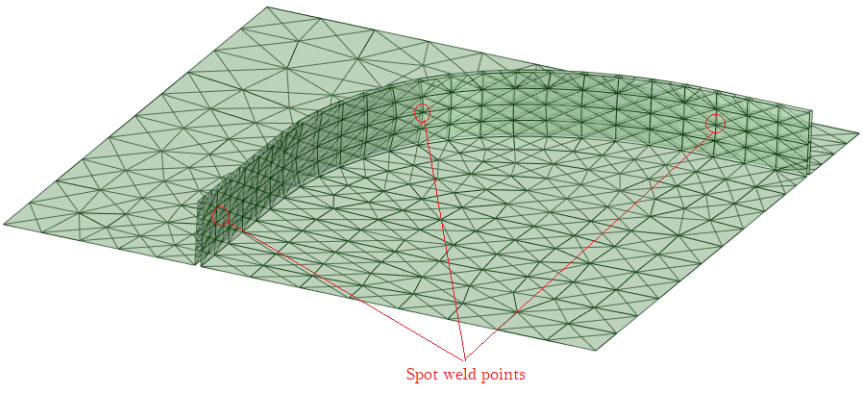


Fig. 18: C shape model

4. The test case model, which consists of seven-spot weld points, is shown in Fig. 19.

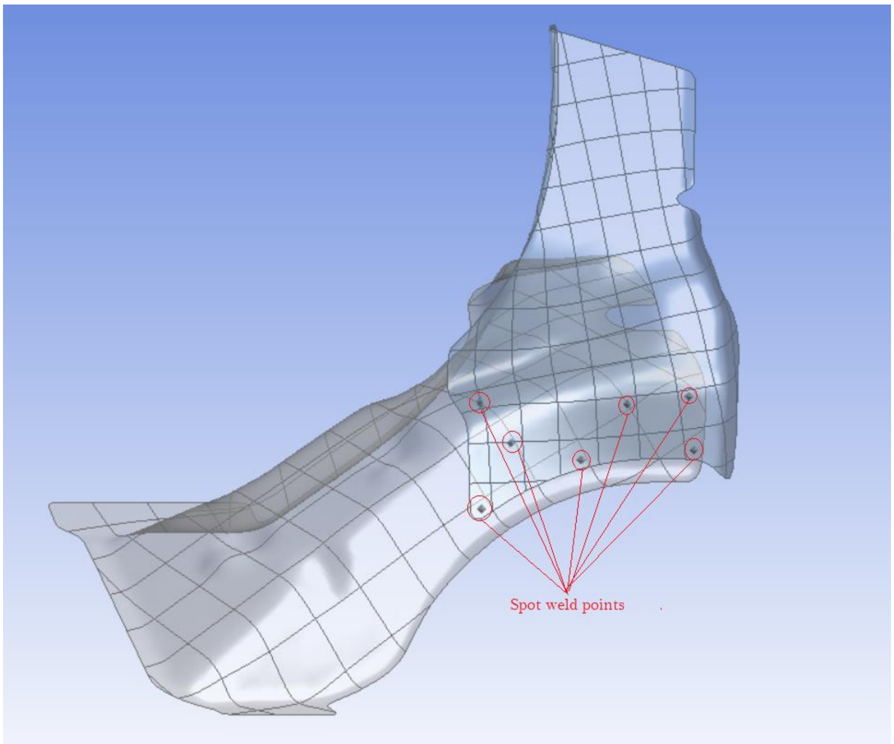


Fig. 19: Test case model

5 Non-rigid variation simulation

5.1 Introduction

The geometrical quality of sub-assembly is measured by using the variation simulation method. As discussed earlier in Chapter 2 geometrical quality can be affected by a number of influencing factors. The advantages of using Non-rigid variation simulation over rigid is that the parts are allowed to bend, the ability to over constraint parts (Add support points), and increased accuracy in simulation.

One of the important aspects of this thesis is to minimize the geometrical variation in subassemblies caused by the joining process (spot weld), which is one of the influencing factors of geometrical quality. The factors involved in joining processes are position, the sequence of spot weld, weld force and influence of heat through weld gun [3]. The non-optimal position of weld points will affect the functional requirements of the subassembly, which in turn increases the development cost [3]. To overcome these challenges, non-rigid variation simulation is performed in the early stages of the product development process.

5.1 Positioning system

The positioning system refers to locator points to support part and fixture (local and target). During the joining process, the parts are locked in six degrees of freedom using locating points to minimize the variation. The position of these locating schemes is also one of the influencing factors of geometrical quality [3]. The positioning system used for the test case is 6 direction, where the first three points (A1 B1 C1) representing a plane which locks two rotation and one translation for the part, the next two points (A2 B2) representing a line which locks one rotation and one translation for the part, the last point (C1) locks the final translation for the part. These positioning systems are provided separately for each part with their corresponding fixture. In non-rigid variation simulation due to its ability to overconstrain its parts, the support points (S1) can be added to minimize the variation. The positioning system for the test case can be seen in Fig. 20. The positioning system is fixed throughout the whole optimization process since the thesis focuses on variation caused only due to Joining Process.

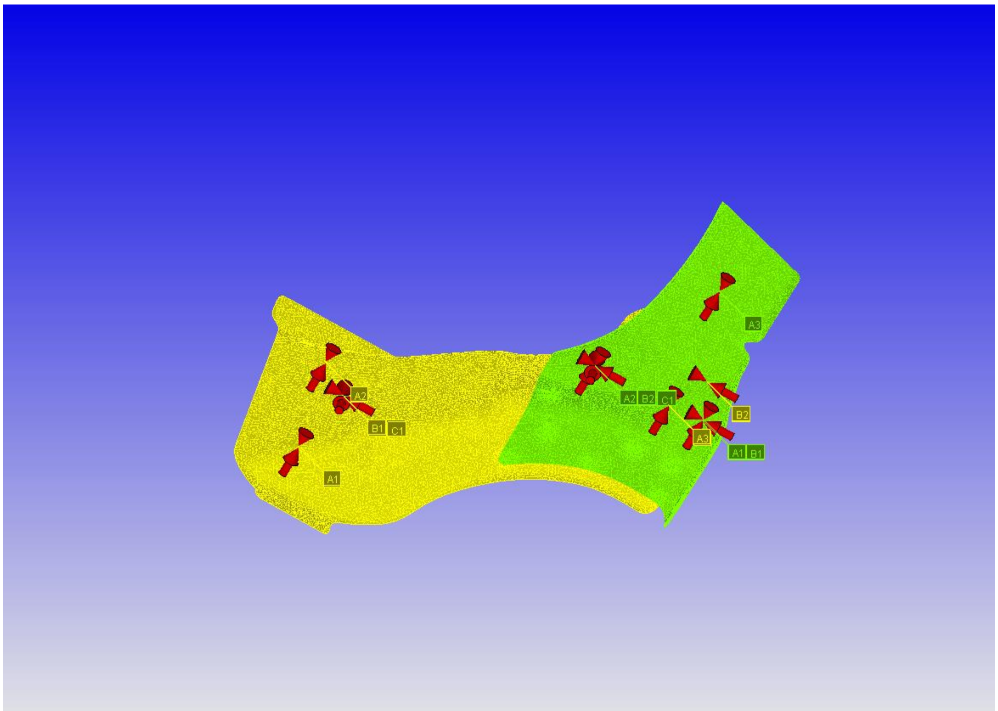


Fig. 20: The positioning system (6 direction) for the test case model

5.2 Creation of subassembly

The subassembly is created to define the parts to be assembled together using one of any joining process. The subassembly also consists of a positioning system, but it is the same as for the part with local and target frames. The support points added for the part is not considered during the subassembly process. Subassembly also consists of the definition of weld points and contact points for the parts to be joined. The definition of weld points and contact points will be discussed in the next chapter. The creation of the subassembly process is shown in Fig. 21.

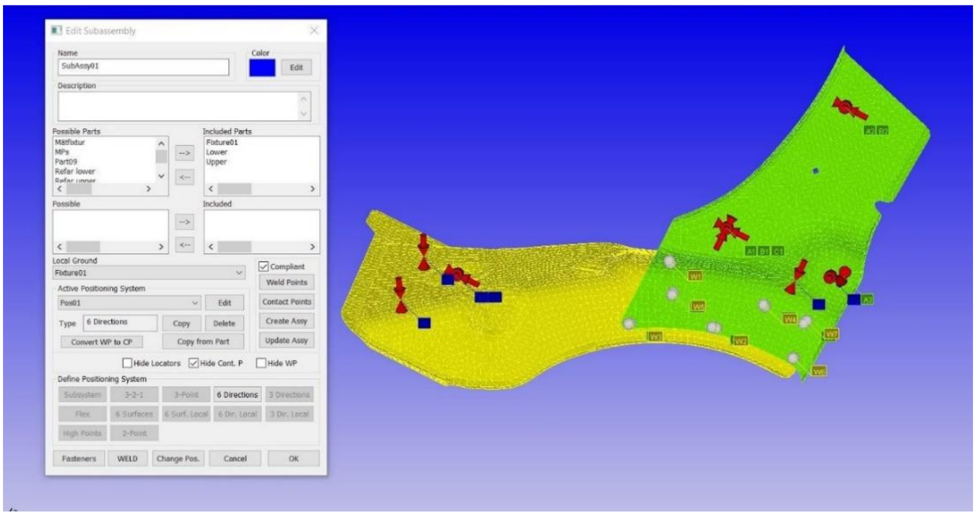


Fig. 21: The subassembly definition in RD&T

5.3 Creation of weld points

The parts are joined using the resistance spot welding method (Balanced gun) which requires both local and target points with their directions. The joining points have three parameters to be considered are position, sequence, and a number of points. The nominal weld points' positions for the test case model are defined with both local and target points. The direction of weld and weld tolerances at each point is also defined based on the model. The test case model in total consists of seven weld points, the change in position, number and sequence of weld points will have an effect in geometrical quality. Previous studies have shown that the determined position of spot weld varies within ± 10mm from the nominal position [1]. In order to study this variation, the position of a weld is moved from the nominal position to all four directions within 1mm and 5mm of distance as shown in Fig. 25. The part variation simulation results with respect to the position of a spot weld are discussed in the results and discussion chapter. The weld point definition in RD&T is shown in Fig. 22.

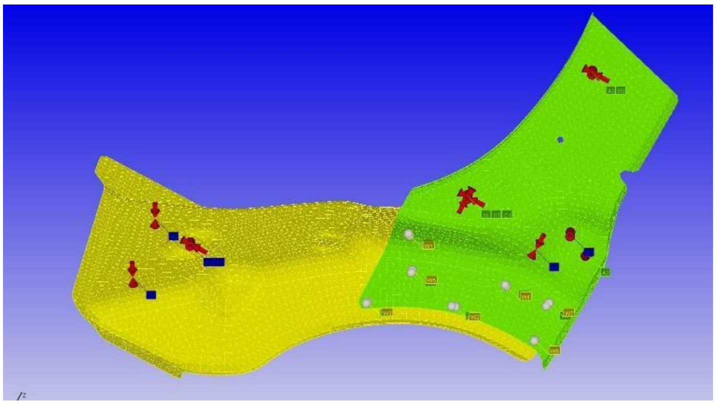


Fig. 22: Creation of weld points

5.4 Creation of contact points

The contact points are defined around the weld point in order to avoid the parts penetrating each other. The contact points are modeled automatically within the RD&T platform for both local and target frames. Each time the position of the spot weld is changed, the contact points should be updated to perform variation simulation. The contact points defined for the test model can be seen in Fig. 23.

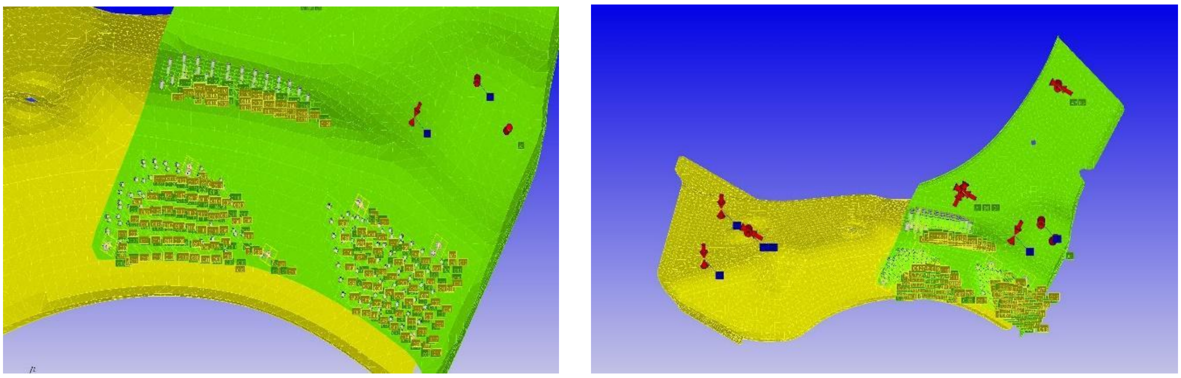


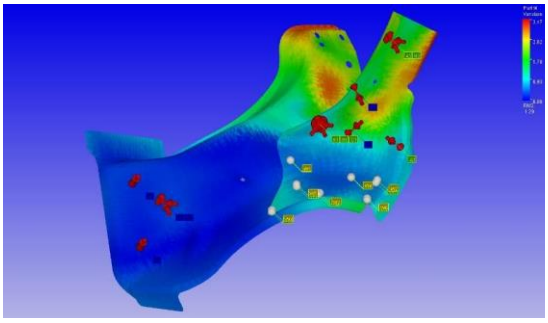
Fig. 23: Contact Modelling for test case model

5.5 Part variation Simulation

Part variation simulation is performed to measure the variation of sub-assembled products with respect to part positioning and position, sequence and number of weld points. The main objective here is to minimize the variation and increase the geometrical robustness in order to reduce the development cost.

Firstly, part positioning is kept constant by changing the position of spot weld, as shown in Fig. 25. This is done to get an overall view on the effect of variation by changing the position of the spot weld. The part variation and RMS values for different positions of spot weld within 1mm and 5mm are shown in Table 1. The part variation for a nominal weld position can be seen in Fig. 24.

Secondly, the part variation simulation is also performed with respect to the sequence of spot weld which is also one of the influencing factors for geometrical quality. Each time weld order is changed (For example (1-2 to 2-1)) and the new model is saved. These different weld order models were then subjected to part variation simulation and their results can be seen in Table 2. This individual simulation is performed to study the effect of part variation with respect to a sequence.



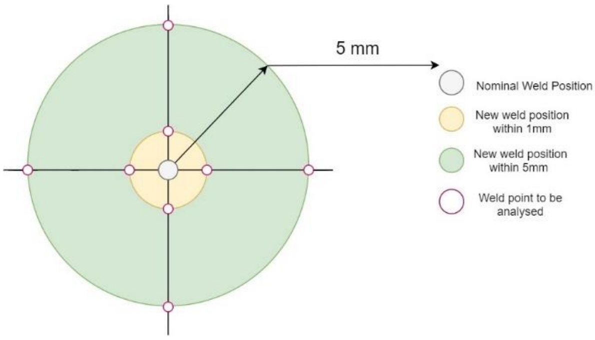


Fig. 24: Part variation on nominal weld position

Fig. 25: New position for spot weld

5.6 Results and discussion

The variation simulation result of both the position and sequence of spot weld will be discussed in this chapter. The test case consists of seven-spot welds in total. The position of each spot weld is changed from the nominal position to 1mm and 5mm in the distance, and each time the contact points modeled are updated. The new position is selected to see the geometrical variation in both upper (5mm) and lower bound (1mm). Firstly, the results of one spot weld position within 1mm distance are listed in Table 2.

Distance In (mm)	Node name	RMS
1	Node A	2.13
1	Node B	2.44
1	Node C	2.42
1	Node D	1.988
5	Node E	4.004
5	Node F	2.118
5	Node G	2.54
5	Node H	3.97

Table 2: Part variation with respect to the location of spot weld

6 Strength Analysis

6.1 Introduction

The location of spot weld selected based on geometrical outcome should also fulfill strength requirements. The strength of BIW Component ensures resistance towards deformation upon mechanical load. After the Welding process, a load is applied to remove the component from the workstation. This effect of loading on the component is studied through this analysis. To perform the simulation Ansys Workbench software 19 is used. The procedure involved in performing the strength analysis are

Step 1: Firstly, the STL assembly component is imported into Ansys Workbench software.

Step 2: The material of the respective component is defined.

Step 3: The spot weld points are modeled based on the coordinate system (XYZ).

Step 4: The geometry has meshed with the Size of mm and 1mm around Spot the weld points.

Step 5: The degree of freedom of the component is locked using Nodal Displacement.

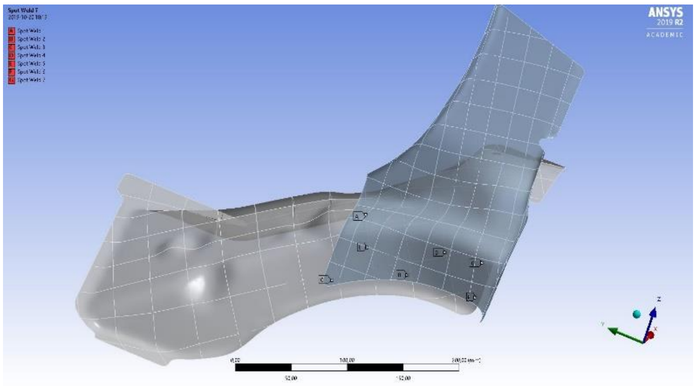
Step 6: Fixed support is established along with two faces of each part.

Step 7: A static load is applied to the component in order to remove the assembly from the fixture.

Step 8: The total deformation of the component is studied through this analysis.

6.2 Creation of spot weld points

The spot weld points are created on the faces of the respective components. Spot weld points can be created using two different methods: the mesh independent and mesh dependent method. The difference between these methods is that spot weld points are created before or after geometry has been meshed. The coordinate data of the spot weld positions are created in a text file as shown in Fig. 27. This file is imported in Ansys and the spot weld points are automatically created on each face of the component based on provided data.



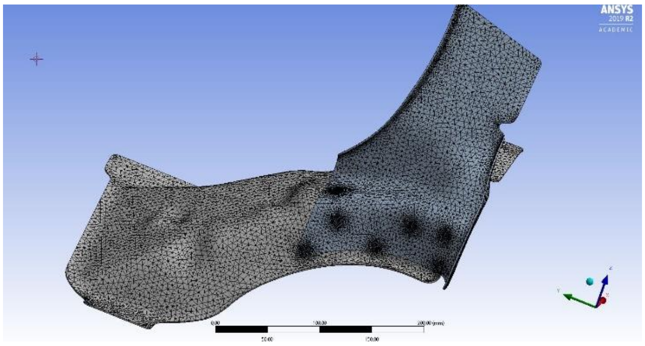

```
# List of point coordinates

# Format is integer Group, Integer ID, then X Y Z all
# delimited by spaces, with nothing after the Z values

#Group 1

1 1 4311.70 -463.20 702.39
2 2 4291.35 -518.52 674.68
3 3 4278.70 -456.95 648.22
4 4 4292.20 -545.03 707.32
5 5 4291.04 -475.85 684.14
6 6 4285.29 -583.76 681.98
7 7 4190.72 -302.62 546.49
```

Fig. 26: Spot weld definition

Fig. 27: Weld coordinate file

6.3 Geometry mesh

The overall geometry is meshed with a size of 3mm. The objective of the thesis is to study the changes when the location of a spot weld is changed within 1mm. Therefore the mesh size is defined as 1mm around spot weld points. This is done by creating a sphere of influence of size 5mm around the weld point. Using this sphere of influence, the mesh size is refined. The mesh refinement around the weld point is shown in Fig. 29 and Fig. 30.



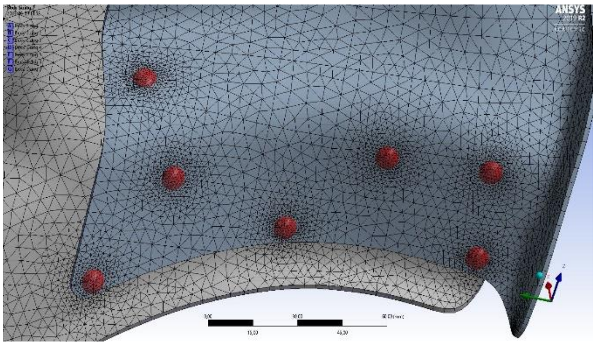


Fig. 29: Mesh definition

Fig. 30: Sphere of influence

6.4 Fixed support and loading condition

Fixed support locks a node, edges or faces in all X, Y and Z direction (zero translation and zero rotation) at the supported end. Fixed support is provided at both ends of the geometry since it is placed on a fixture to restrict any motion during the welding process.

After the welding process, a small load is applied to the subassembly before going into the final assembly. In every case, the load applied is very small, but it can make differences in the overall strength of the final assembly. Hence, this small load is included in the boundary condition as Force load.

6.5 Total deformation:

The welded subassemblies because of a small load undergo a deformation before going into the final assembly. Therefore, changes in the total deformation values of a subassembly with respect to the location of spot weld points are studied through this simulation. The deformation result with respect to one particular spot weld point is shown in Fig. 31.

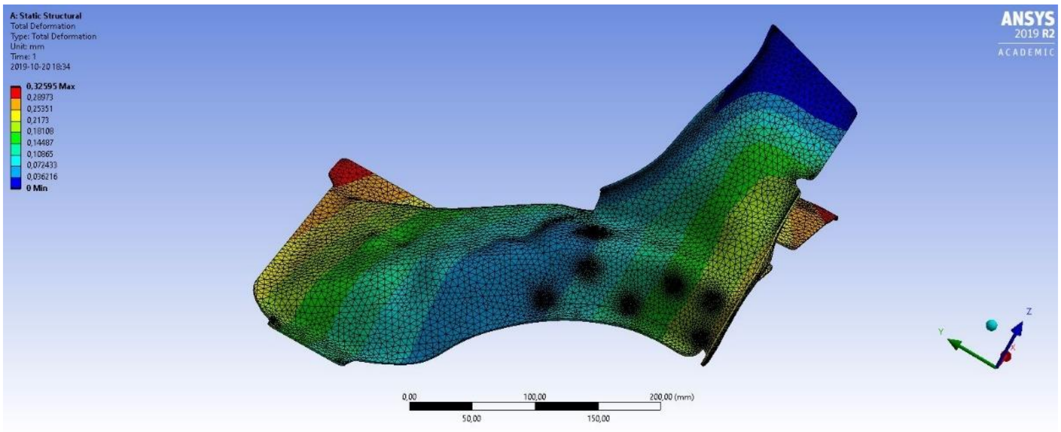


Fig. 31: Simulation result of nominal weld position

6.6 Individual simulation strategy:

As discussed earlier, the location of spot weld has huge influence over geometrical quality, Stress, and Strength of a subassembly. To understand the effect of changes in the strength of a subassembly concerning spot weld position is studied. The study is done by changing the position of spot weld from a nominal position to 1mm and 5mm in four direction as shown in Fig. 32. In order to perform the simulation with different spot weld positions each time, the spot weld coordinate file is updated with new X, Y and Z values. Later during optimization, this process is automated with the use of interaction between MATLAB and Ansys. The deformation values obtained for different spot weld position is discussed in the next chapter.

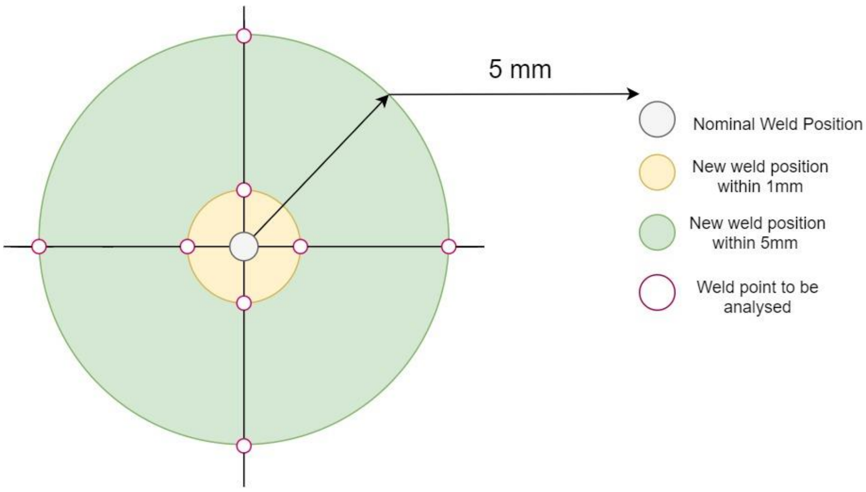


Fig. 32: Simulation strategy for individual discipline

6.7 Results and discussion

This individual-level simulation result gives a clear understanding of changes in the overall strength of a subassembly with respect to the position of the spot weld. The deformation result of different spot weld positions within 1mm and 5mm from the nominal position is shown in Table. 3. From the values, it can be clearly seen that changing the position of spot weld up and right from the nominal position shows a better overall strength. Also, changing the position of spot weld towards the edge of a subassembly shows a poor strength quality. These two results were valuable in identifying the best strategy for setting up the multidisciplinary optimization problem.

Distance in (mm)	Node name	Total deformation (mm)
1	Node A	2.146
1	Node B	2.167
1	Node C	2.150
1	Node D	2.151
5	Node E	2.106
5	Node F	2.171
5	Node G	2.205
5	Node H	2.1542

Table 3: Simulation results for individual discipline

7 Residual stress

7.1 Introduction

During the spot-welding process, the spot weld electrode comes in contact with the components where significant heat is applied to a predefined position. This position of spot weld affects the residual stresses in which the components expand or contract with respect to temperature difference. The temperature difference is when the spot weld positions are heated up to 1400°C, i.e., standard spot weld temperature [21] for the joining process and the room temperature of the overall part. Like the above two disciplines, residual stress analysis was conducted to study how changing the position of spot weld affects the residual stresses developed in the component. To perform this study, Ansys workbench 2019 is used. The steps involved in performing residual stress analysis are

1. Import the STL model and define the spot weld temperature zone

2. Geometry mesh

3. Degree of freedom of the component is locked using nodal displacement

4. Fixed support and loading condition

5. Perform thermal analysis

7.2 Spot weld temperature zone

The temperature zone represents the position and thickness of the electrode. The type of spot weld gun chosen is a balanced type that consists of two electrodes that come in contact with both the upper and lower part of the test case model. Since the nominal position of the spot weld electrode is well-known already, the next step was to find a suitable thickness. From the standards of spot welding, the thickness of the electrode depends upon part thickness, as shown in Table. 4 [22]. According to the standards, the suitable electrode thickness for the test case model is chosen as 5mm. With this in mind, the spot weld temperature zone at a nominal spot weld position with a diameter of 5mm is created for both the parts of the test case model are shown in Fig. 33.

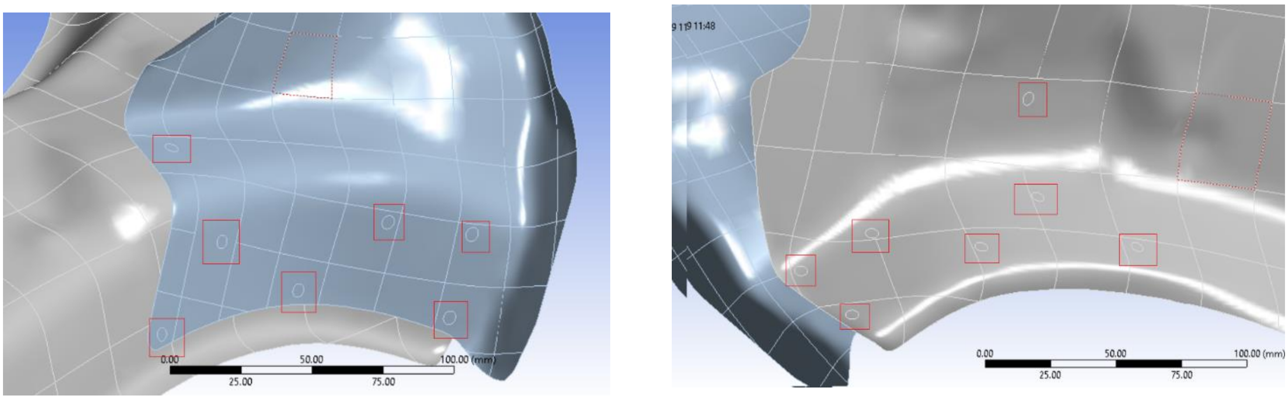


Fig 33: Spot weld temperature zone

Thickness Range from (mm)	Thickness Up to (mm)	Electrode Diameter (mm)
0.4	0.6	4
0.6	0.8	4
0.8	1.0	5
1.0	1.2	5
1.2	1.6	6
1.6	2.0	7
2.0	2.5	8
2.5	3.0	9

Table 4: Standards in the selection of electrode diameter [21]

7.3 Geometry mesh

The overall geometry has been meshed with a size of 5mm. Since the objective of this thesis is to study the changes when the position of a spot weld is changed from 1mm to 5mm in all directions. Hence the spot weld temperature zone is selected and defined with a mesh size of 1mm. The mesh refinement around the weld points is shown in Fig. 34.

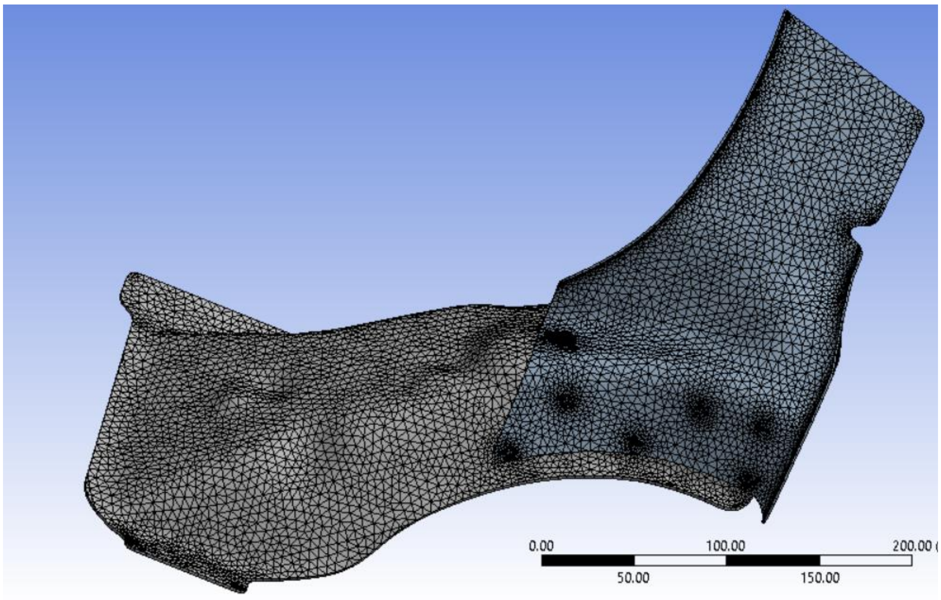


Fig. 34: Geometry mesh

7.4 Degree of freedom

The position of components in space has six degrees of freedom which consist of three translation and three rotation in total. Hence, the components are locked in every degree of freedom to spot weld the components without any variation. Nodal displacement is used to model this in the Ansys platform. To model this in Ansys, a node is selected and locked in X, Y, and Z directions. The data for applying nodal displacement to the respective node is taken from the RD&T model. Fig. 35, shows the boundary condition for nodal displacement.

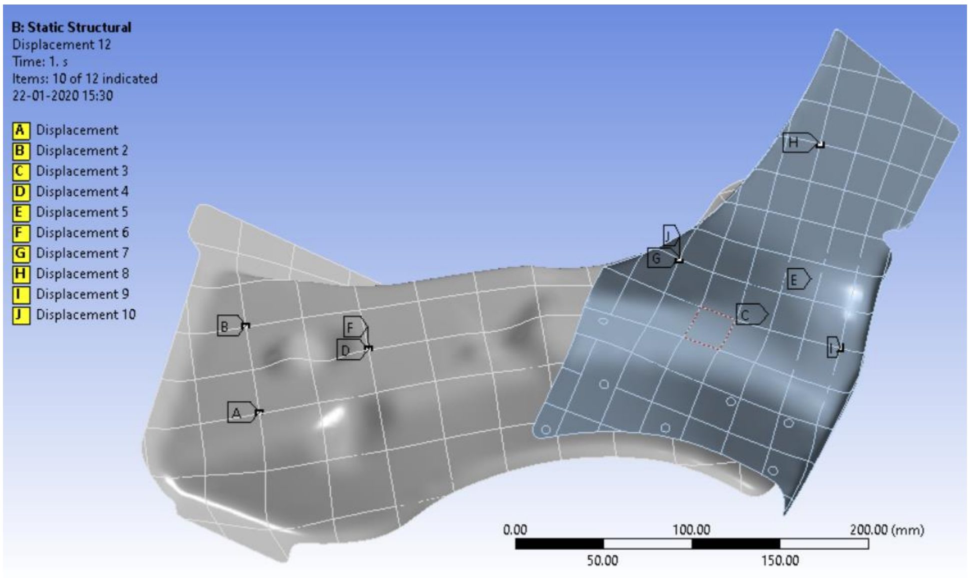


Fig. 35: Degree of freedom

7.5 Fixed support and loading condition

Fixed support locks a node, edges, or faces in all X, Y, and Z direction (zero translation and zero rotation) at the supported end. Fixed support is provided at both ends of the geometry since it is placed on a fixture to restrict any motion during the welding process.

During the welding process, the required spot weld temperature is applied to a zone that is created during the first step of this analysis. For this analysis, a standard spot weld temperature of about 1400°C is applied to every temperature zone that is created, and the overall part is subjected to room temperature, which is 24° C. This temperature definition in Ansys is shown in Fig. 36.

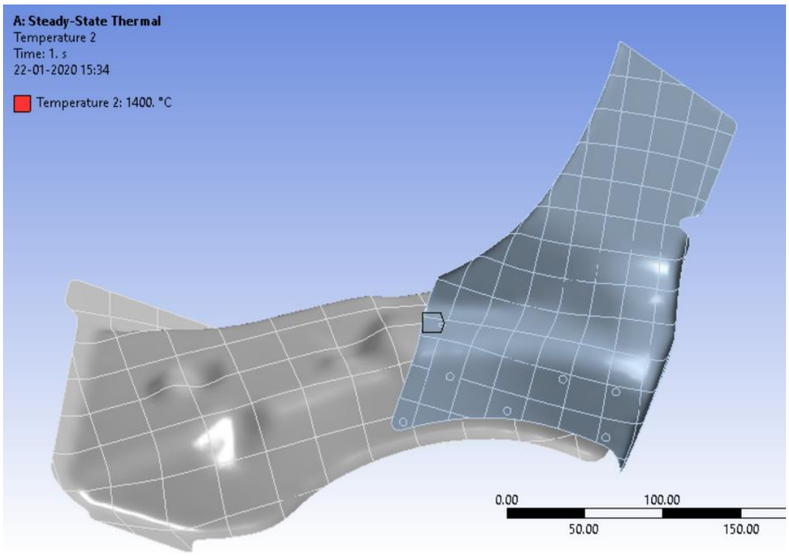


Fig. 36: Temperature definition

7.6 Residual stress analysis

The large temperature difference created by heating the part to join and allowing it to rapidly cool down causes heterogeneous deformation thus resulting in residual stress. These residual stresses of a subassembly with respect to the position of the spot weld is studied through this simulation. The residual stress result with respect to a nominal weld position is shown in Fig. 37.

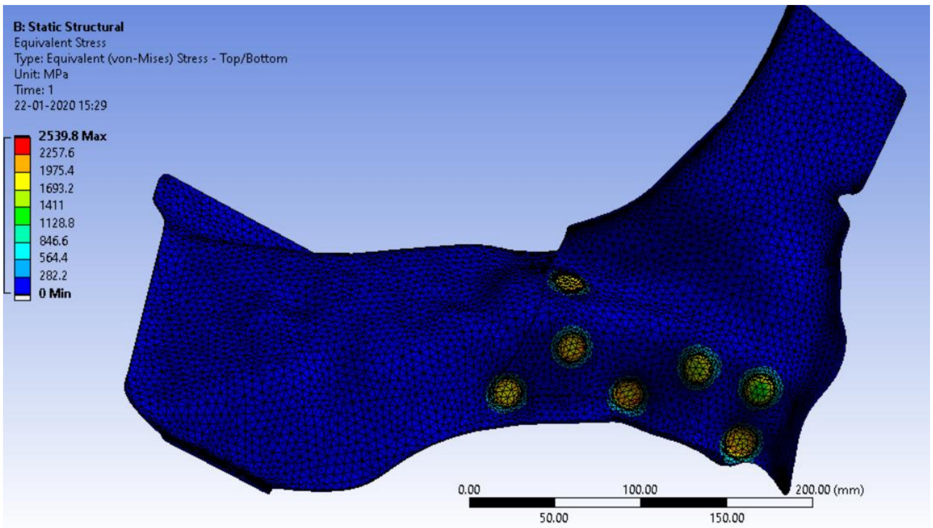


Fig. 37: Residual stress analysis

7.7 Individual simulation strategy

To understand the effect of changes in residual stress of a subassembly with respect to spot weld position is studied. The study is done by changing the position of spot weld from a nominal position to 1mm and 5mm in four directions as shown in Fig. 25. In order to perform the simulation with different spot weld position each time, the position of spot weld temperature zone that is created during the first step of this analysis is to be changed. To do this, the parametrization technique from Ansys is used. The position of each temperature zone i.e., X, Y, and Z are parameterized and can be updated with new values within 1mm and 5mm once for all as shown in Fig. 38. Later during optimization, this process is completely done within Ansys, and the results are provided as a text file. The residual stress values obtained for different spot weld position is discussed in the next chapter.

Table of Design Points

	A	B	C	D	E	F	G	H	I
1	ne	P2 - weld1x	P3 - weld1y	P4 - weld1z	P5 - weld2x	P6 - weld2y	P7 - weld3x	P8 - weld3y	P9 - weld3z
2	nits	mm	mm	mm	mm	mm	mm	mm	mm
3	Current)	4318.7	-463.2	702.39	4291.4	-518.52	4278.7	-456.95	648.22
4		4312	-461.59	705.08	4291.4	-518.52	4278.7	-456.95	648.22
5		4311.3	-462.61	705.27	4291.4	-518.52	4278.7	-456.95	648.22
6		4314.4	-463.58	706.25	4291.4	-518.52	4278.7	-456.95	648.22
7		4315.1	-462.78	706.13	4291.4	-518.52	4278.7	-456.95	648.22
8		4310.6	-463.02	705.27	4291.4	-518.52	4278.7	-456.95	648.22
*									

Fig. 38: Parametrization of spot weld temperature zone in Ansys

7.8 Results and discussion

The residual stresses for different spot weld positions within 1mm and 5mm from the nominal position are shown in the table.

Distance in (mm)	Node	Residual stress (pa)
1	Node A	2.07e+09
1	Node B	2.11e+09
1	Node C	2.13e+09
1	Node D	2.05e+09
5	Node E	2.04e+09
5	Node F	2.22e+09
5	Node G	2.27e+09
5	Node H	2.08e+09

Table 5: Results of residual stress

8 Prerequisite for Multidisciplinary Optimization process

8.1 Interactive link between MATLAB and Ansys

This chapter is focused on the method developed to establish a connection between MATLAB and Ansys to perform multidisciplinary optimization. This chapter is divided in two parts, where the first part describes the requirements to establish the connection and the method itself in the final part.

8.1.1 Introduction

In general, the design of products requires interaction between several disciplines to reach a common point that satisfies both system and sub-system requirements. As discussed earlier, the chosen disciplines for this thesis are geometrical variation, residual stress, and strength analysis. Matlab tool is used in this thesis which not only supports Design Optimization but also links all the disciplines in one place. A method that will establish a platform that integrates multiple software to optimize the Product design over a number of iterations to achieve system requirements. This platform is then used to establish a connection between all the software such as Ansys Mechanical Apdl (residual Stress and strength analysis), RD&T (geometrical variation), and Matlab (design optimization). This connection will help understand how a design variable affects every different discipline.

To establish a connection between Ansys Mechanical Workbench and Matlab was one of the objectives. The different requirements starting from Software selection to achieve the desired result were to be framed before developing a platform. This requirement throughout the thesis allowed to foresee the future and in fact sometimes helpful in keeping the simulation technique within the bounds. One good example would be during the software selection, some of excellent FEA software has to be ruled out because it should not only support for Individual simulation but also during MDO (Overall simulation) to be able to establish a connection. From other perspectives, it is either difficult or expensive (Software license) to establish a connection. After going through each requirement with respect to commercial FEA Software available on the market, the team decided to move forward with two software such as Abaqus (strength analysis) and Ansys Workbench (thermal analysis) which satisfied all the requirements.

Once the software was selected, the literature study was performed to understand the method available to establish a connection between Abaqus, Ansys Workbench and Matlab. Concurrently, Boundary condition for Individual Discipline simulation was developed in their respective selected software platform to be able to combine every discipline in the final simulation. After understanding the method available to establish a connection, there were quite a lot of other aspects to consider, for example, memory requirements, RAM allocation which would play a major role in final simulation (MDO). Therefore, important learning through this study that increasing the number of software platforms would increase the complexity of the problem in establishing the connection. Hence, Ansys Workbench which supports different packages such as structure, thermal, Fluid will be a suitable option to be used for both the disciplines such as Thermal stress and Strength analysis. Finally, the number of different software was reduced, and thus decreasing the complexity of the problem.

The different requirements that are considered to select suitable software packages for this thesis are shown in Table 6.

NO	REQUIREMENTS	SCORE (Priority)
1	Suitable not only for Individual simulation but also for overall simulation (MDO)	5
2	Software License (Available through Chalmers and Winquist Laboratory)	5
3	Method availability to establish a connection in Matlab	4
4	Computational Time and Cost	2
5	Memory requirements and RAM Allocation	4
6	Able to edit (Change Design variable) from the initial boundary condition – Degree of flexibility	4
7	Ability to run the software through Matlab.	5
8	Resources to learn and read about the software	3

Table 6: Requirements list for software selection

These were initial requirements formulated during the software selection, which ultimately be helpful in every other stage in this thesis. The selection was made by passing all different software packages through the above-listed requirements. Initially, the work started with the software's in which the team felt that is comfortable to work with or with the previous experience. During this process, there was a wide scope to work with and became a trial and error method to determine the software that is suitable to establish a connection. At the time, the selected software would work with some cases but not completely. For example, we have chosen Simufact welding software package to perform thermal analysis, which is an easy and efficient platform to do these types of analyses. The team started reading about the software and tried to acquire a student license but then realized in the end, it is difficult to establish a connection with the Matlab based on our requirements. With this knowledge, a requirements list is created to be able to use it for the selection of the software. This requirements list helped save a lot of time in coming back and forth to the same place that is choosing the software that would satisfy all the requirements.

During the process of framing the requirements, not all the requirements are identified at the same time. For example, able to edit (Change Design variable) from the initial boundary condition – Degree of flexibility from the software is required to be able to change it easily through a simple code in Matlab during the multidisciplinary optimization process. If the software does not allow to do so, a completely new boundary condition as discussed in strength analysis chapter needs to be defined during each iteration, which will, in turn, increase the computational cost and time and complexity of the problem. These are the process involved in creating the requirements for the software selection to be able to use for multidisciplinary design optimization.

8.1.2 Method used to establish a connection between Ansys Mechanical Apdl and Matlab

This chapter will be focused on the method or platform itself used to establish the connection between Ansys Mechanical Apdl and Matlab.

To solve the design problem using multidisciplinary design optimization requires a lot of interaction between each discipline at both system and sub-subsystem level. A method that will connect all the disciplines in one place needs to be developed. As discussed in the previous chapter, Matlab is used to bring every discipline that is geometrical variation, thermal stress and strength analysis. Each time the MDO algorithm generates a design variable (position of spot weld points), all the disciplines should be able to read the variable and create a response for it. Response for each discipline is the part variation (RMS) in geometrical variation, thermal stress in thermal analysis and total deformation (strength analysis). This will work if a platform in MATLAB would there to execute multiple software.

A literature study was done to see the methods available to establish a connection between Ansys Workbench and Matlab. From this study, we have found that two direct ways that could be used to establish the connection. The first method is using the AAS toolbox which establishes a connection between Ansys Workbench and Matlab directly. To use, this toolbox requires Ansys access to the customer Portal. All the design variable or boundary conditions here can be established or changed using a Matlab script. The second method is creating a model in Ansys Mechanical Apdl along with the necessary boundary condition. These models in Ansys Mechanical Apdl contain Macro code which was taken to Matlab even here the design variable or boundary condition can be established or changed in Macro code.

Initially, the team worked with both methods to see which one is more suitable for this thesis. Each method has its own advantages and disadvantages, for example, the AAS toolbox method required a lot of understanding on a programming level in changing a particular boundary condition (Spot Weld Points) through a script in Matlab. Whereas the second method was quite a direct method by just changing the node number manually each time in Apdl Code. Later, a Matlab script that automatically updates the new weld point based on the node number from Ansys was developed. This will be discussed in detail in the next chapter. Finally, the second method was chosen to move forward for a multidisciplinary optimization method.

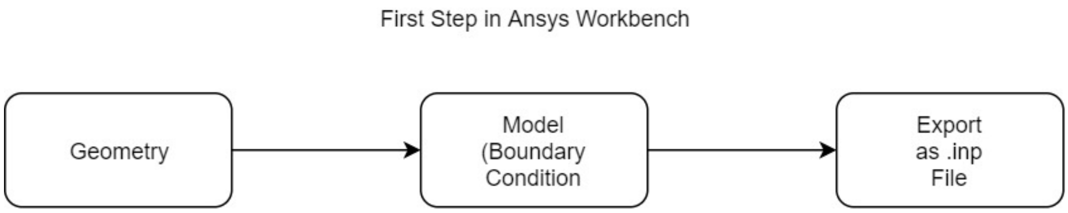


Fig. 39: Step 1 involved in establishing the connection

Before this method can be used to establish a connection between Matlab and Ansys Mechanical Apdl, it requires two minor steps to be followed. The geometry is modeled with all necessary boundary conditions in Ansys Workbench 2019, which is a simple interface and easy to use. This modeled geometry is converted as .inp files (write input files) in Ansys Workbench, which can be imported in Mechanical Apdl to solve the problem. Once the model is solved, it is then ready to be taken to Matlab as a .BAT file to establish a connection between both the software.

The second step involved before establishing the connection is

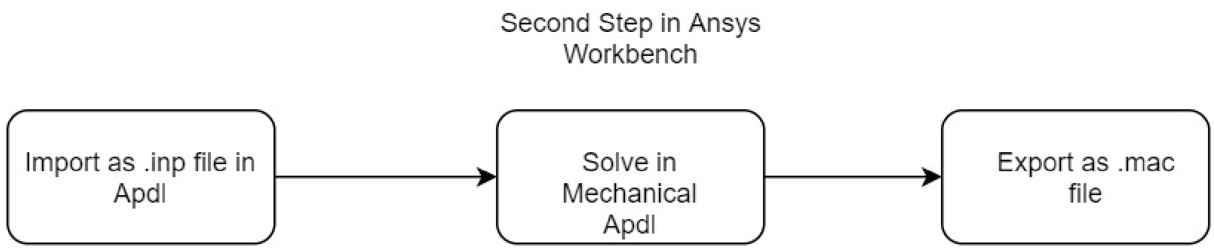


Fig. 40: Step 2 involved in establishing the connection

Once the two minor steps are performed, the next step will be establishing a connection between two software. Three documents are required to start the connection such as Macro file from the Ansys mechanical Apdl, Batch file that is a directory link of Ansys mechanical Apdl from the user's computer and Empty text file to write the required output. All these files are required to be in the same directory. Fig. 41 shows the respective file format to establish a connection.

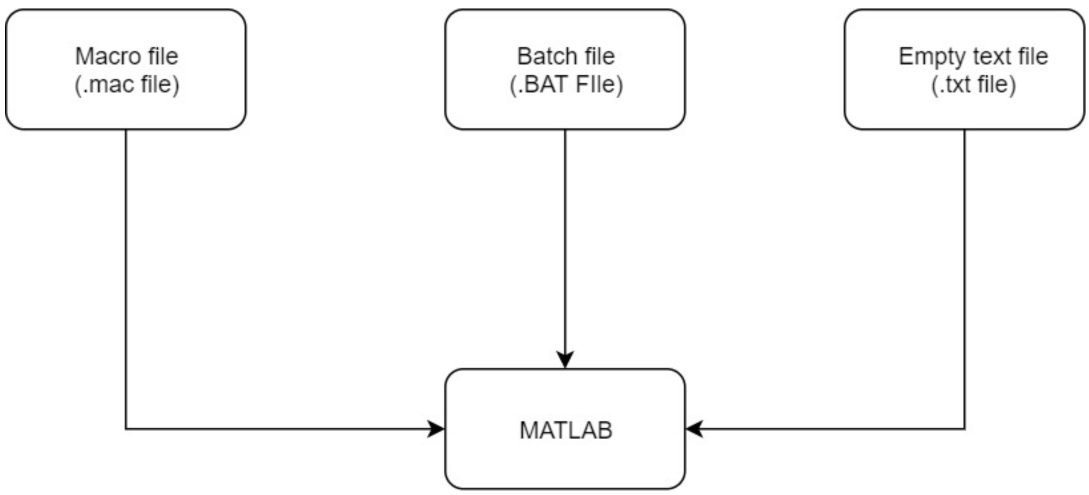


Fig. 41: Final framework to establish the connection

This connection between Matlab and Ansys Mechanical Apdl helped automate the tasks in the multidisciplinary optimization process. The task refers to changing the design variables each time during the iteration. The macro file which contains different information that includes geometry data, mesh data, the definition of spot weld, loading condition, and supports location. A Matlab script was developed which reads all these data, identify the spot weld position and change to a new position each time in a loop until an optimal solution which satisfies both the objective function and constraints is found. This will be discussed in detail in the next chapter. The output generated through this connected simulation contains a lot of information that is not necessary to consider for this thesis. For example, a strength analysis only requires a total mean deformation value from the output file for function evaluation. Therefore, a Matlab script that reads the file and extracts the particular deformation value was created.

8.2 Optimization search algorithm

8.2.1 Introduction

This chapter is focused on the search algorithm developed in Matlab, which was one of the main requirements for the multidisciplinary optimization process. This chapter will be divided into two sections, which describe the idea behind the search algorithm and how this algorithm is used during the MDO process.

The main objective of this thesis to study the position of spot weld with respect to three disciplines such as geometrical variation, residual stress, and strength analysis. In general, when two parts to be joined using a spot weld method, there is always a huge number of positions available to decide upon the best possible spot weld position, which satisfies the requirements. In this thesis, the test case model consists of 20000 possible points around which seven points are to be defined to join the parts which referred to the nominal weld position, representing the allowable position variation. During the optimization process, evaluating all the possible 20000 points with respect to all three discipline are computationally time-consuming and not very cost-effective. An effective search algorithm that will reduce the number of evaluating points but still able to get the optimized result is to be developed.

8.2.1 Search algorithm

The search algorithm concept is shown in Fig. 42. There are different requirements needed to consider in developing this algorithm. As discussed in the previous chapters that the total space constraint is 5mm in diameter from the nominal weld position.

1. The quality of the final result should not be sacrificed due to the reduction in the number of evaluating points.
2. Secondly, the algorithm should be able to cover points in all directions from the nominal weld point.

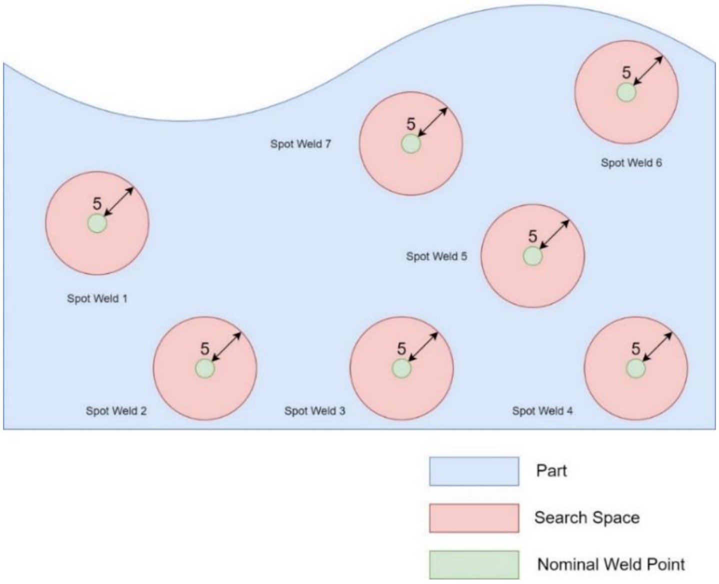


Fig. 42: Search space

According to the requirements, a search algorithm in detail is shown in Fig. 43 is developed. This algorithm is more user-friendly which mostly depends upon distance and direction with respect to the nominal weld position. From the nominal weld position, a search area such as 1mm, 2mm, 3mm, 4mm, and 5mm are defined by calculating the distance from nominal to the respective weld point. The below method shows the step involved to calculate the distance between two coordinate points.

Nominal weld position A = 4311.70 -463.20 702.39

Chosen weld position (from the data file) B = 4291.38 -518.51 674.67

A Matlab tool to calculate the distance between two coordinate points is used which is

Norm(A-B)

This procedure is repeated by comparing the nominal weld position to the rest of the weld point position from the data file which contains all possible weld positions around 5 mm. By doing this, Weld points around 1mm of diameter from nominal are sorted out. During the individual simulation, the team found that changing the weld position to 1mm from the nominal does not make any difference in the result. Therefore, it was not necessary to evaluate all the points within 1mm once the data was sorted during the MDO process. To refine the search area further, the weld points are again sorted based on the direction. This was done by shifting all the data to the origin that is a nominal weld position. By doing so, the weld points that are within 1mm in all four direction as shown in the fig was achieved. The same procedure is repeated for other search areas such as 2mm, 3mm, 4mm, and 5mm. Finally, this algorithm generates two text files such as Sort Dist and Sort Dire which are sorted based on distance and direction from the data file.

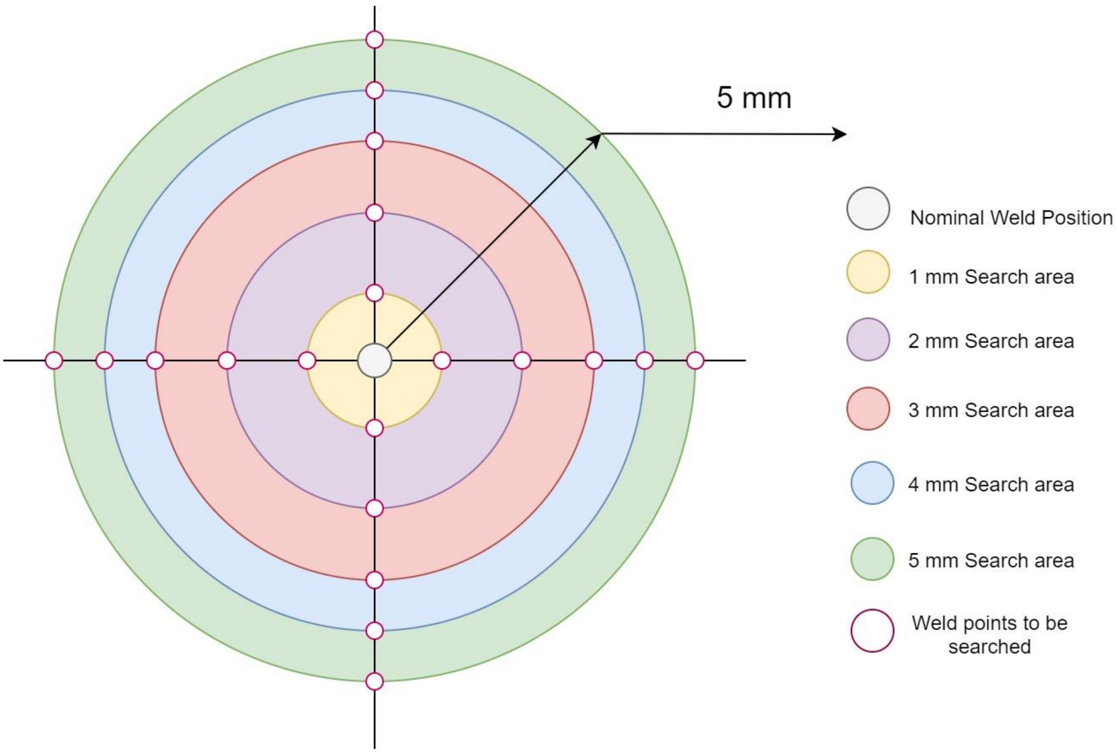


Fig. 43: Detailed view on the search space

8.2.3 Search algorithm in detail

This part of the section will describe the procedure involved in setting up the algorithm as shown in Fig. 44, 45, and 46.

The first stage will concentrate more on the required data from the individual simulation of each discipline.

1. The mesh size used in individual simulation shall be of the same size. The number of nodes and elements shall be equal in all the disciplines. Hence, the mesh file used in Ansys is exported to RD&T to be able to use the same mesh model in every discipline. This would allow changing to the same new position of spot weld (Coordinate point) in all the disciplines during the MDO Process.

2. Once the same mesh model was used in all three disciplines, the next step was to extract the possible weld information from Ansys and RD&T. In RD&T, the weld points that are generated through contact modeling were extracted directly with contains their coordinate points data in terms of the sequence. But, in Ansys, a refined mesh node around 5mm from the nominal position is manually selected and exported directly which contained the coordinate points and their respective node number.

3. From here, it was understood that different types of inputs needed to change the position of spot weld in the respective discipline. For example, in geometrical variation discipline (RD&T), the required input was sequence but in the other two disciplines such as Strength and Thermal stresses (Ansys), the required input was node number. Lastly, the original data file extracted from both RD&T and Ansys was saved as

Original Ansys and original RD&T which will be used in the final stages of this algorithm.

4. Even though the required input is varied for the different disciplines, the common thing for all the disciplines was the coordinate point. This was the reason for this algorithm that developed was based on the coordinate point of the spot weld.

5. Even though the same mesh model was used in all three disciplines, there was a possibility that not all the weld point data from RD&T will match exactly with Ansys weld data in terms of coordinate points. This is based on the definition of weld points in each discipline. For example, the weld points defined in RD&T were based on contact modeling (Automatically created by the software itself) whereas in Ansys a refined mesh around a nominal weld point was manually selected to define the weld points.

6. The obvious next step was to eliminate some position of weld points that were not matched equally in all three disciplines. Without this step, the function evaluation with differences in weld position from all three disciplines would not provide the optimized result.

7. Once all the steps discussed above are satisfied, the next step was to import the similar coordinate points from all the disciplines to Matlab to sort based on distance and direction (Sort Dist and Sort Direc) with a code which is shown in Appendix C.

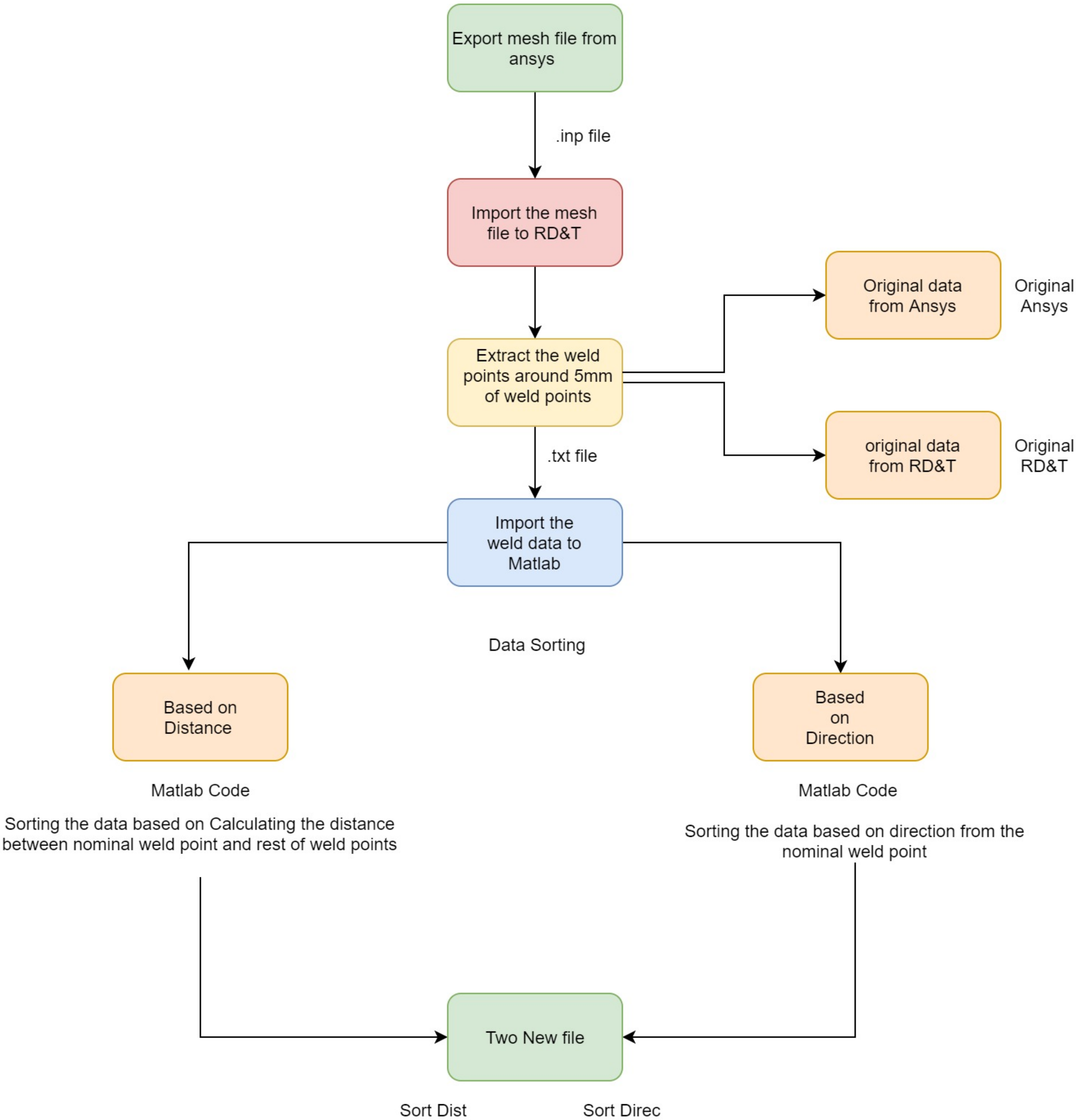


Fig 44: Flow diagram- 1 for the search algorithm

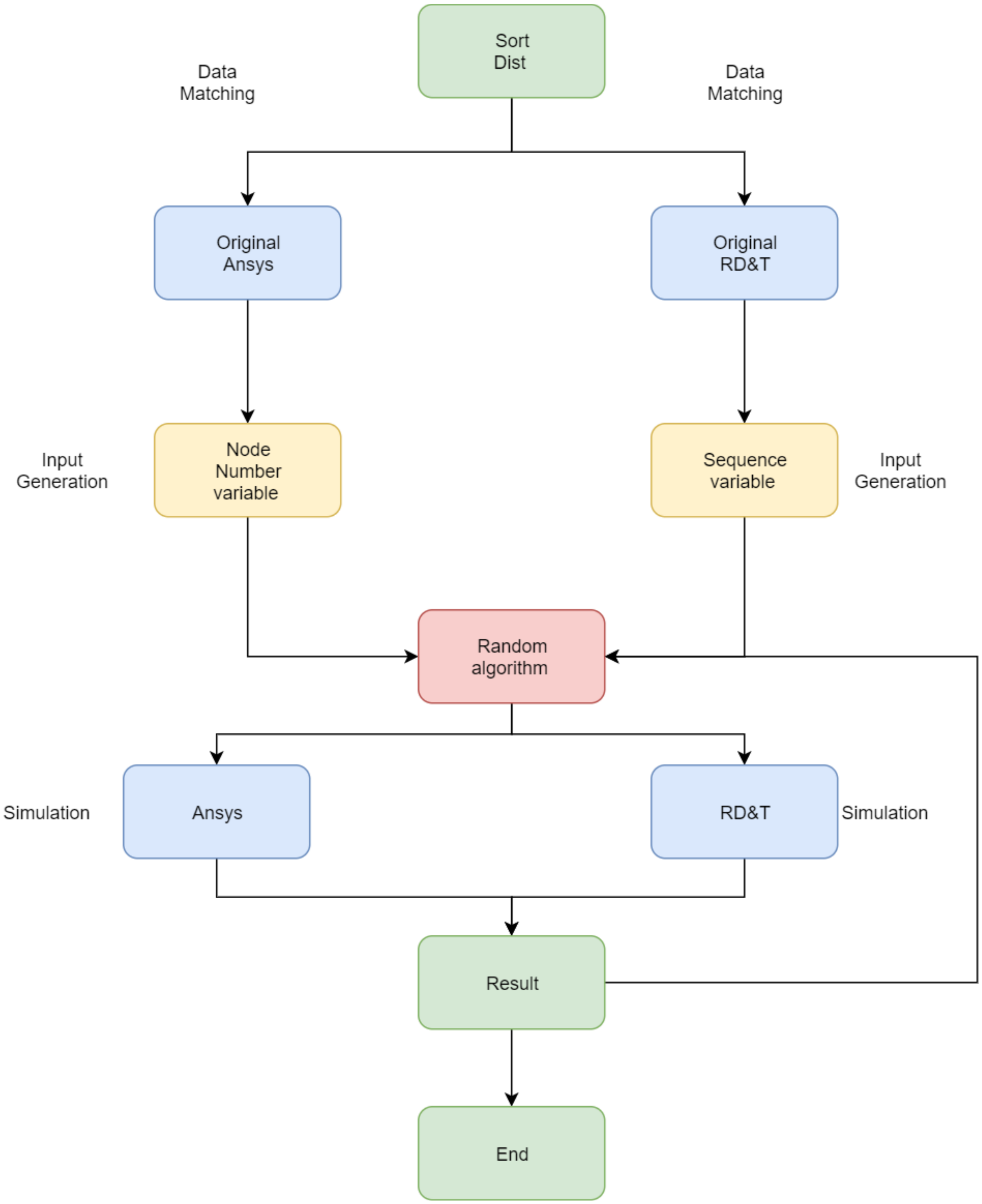


Fig 45: Flow diagram- 2 for the search algorithm

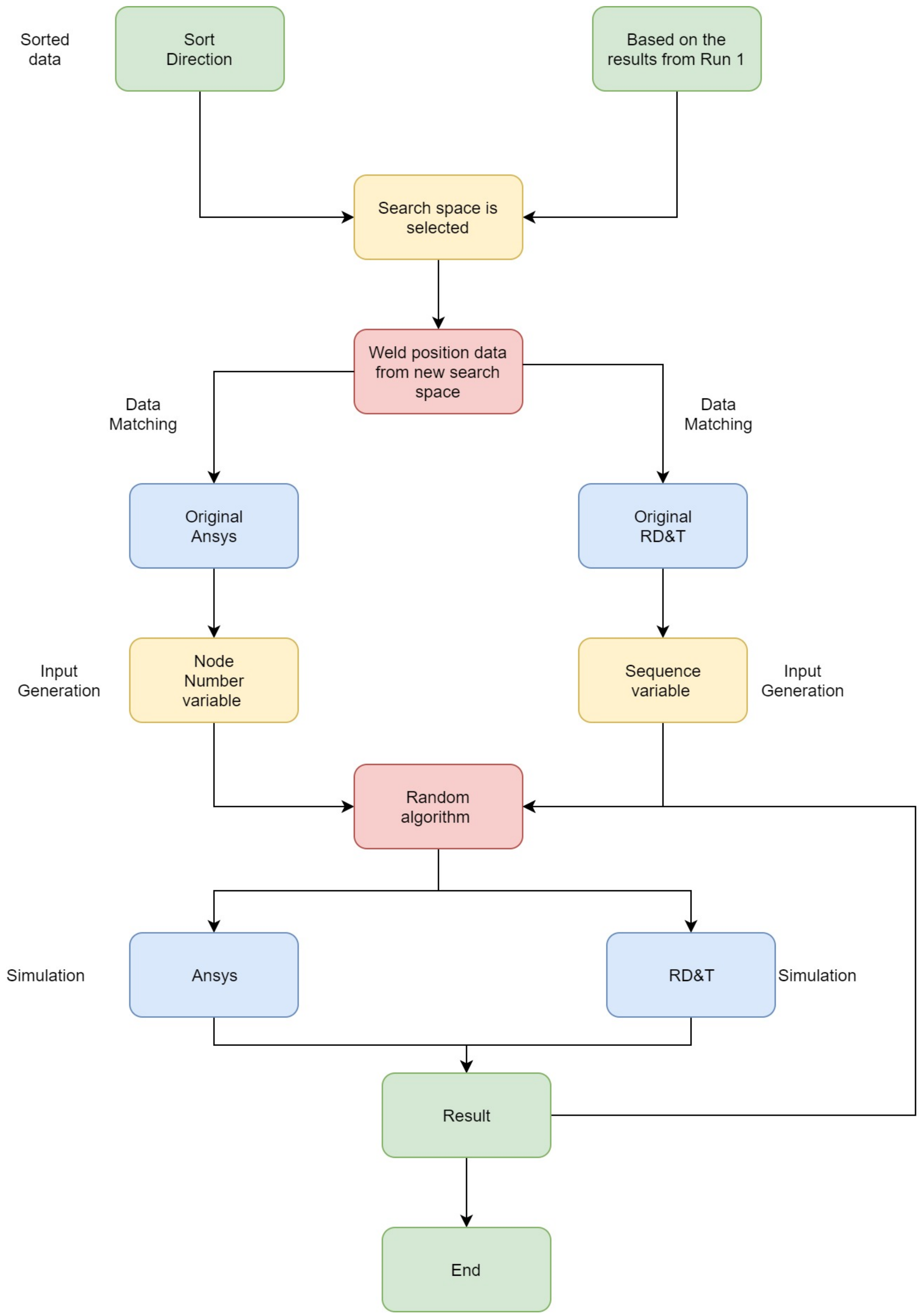


Fig 46: Flow diagram- 3 for the search algorithm

9 MDO Formulation

9.1 Introduction

This section will focus on the method developed to perform MDO. From the literature, there are different types of problem formulation, such as IDF (Individual discipline feasible), MDF (Multidiscipline feasible), and also different types of other methods available to formulate the problem. From these different methods, one architecture that supports this case was used to formulate the problem.

9.2 Multidisciplinary optimization

As discussed in the previous chapter, there are three disciplines, i.e., geometrical assurance, strength analysis and residual stresses that are needed to be considered to choose the optimal position of the spot weld. An MDO method developed to evaluate the positions of spot weld for the test case model is shown in Fig. 47. This method consists of two levels of evaluation namely the subsystem level which is formed based on the IDF principle and the other level is system level. The coming sections will focus on each of the levels in detail.

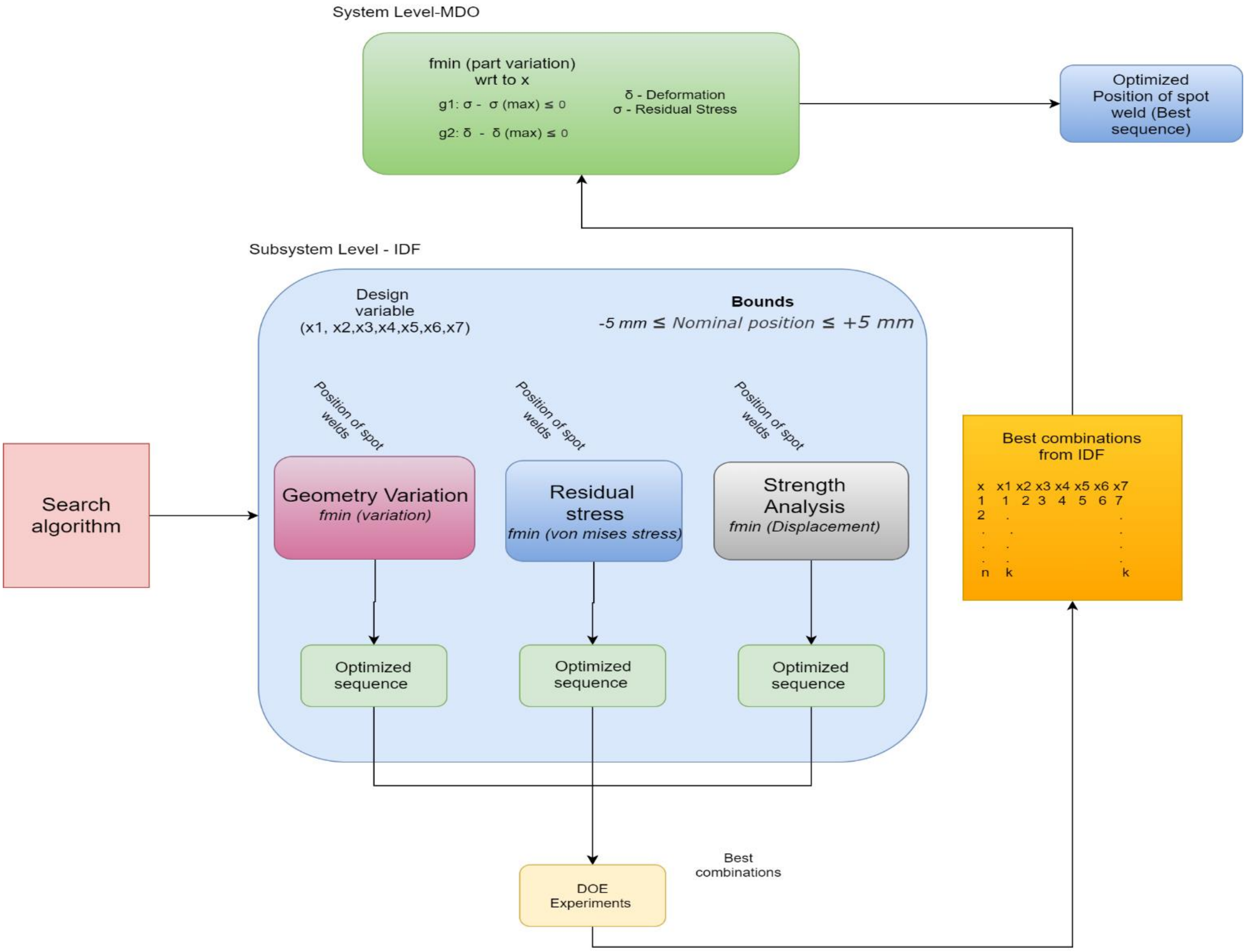


Fig. 47: MDO problem formulation

9.3 Problem formulation

$$minimize \quad RMS(x, y_1(x), y_2(x), y_3(x)) \quad (6)$$

$$subjected\ to\ 'x'$$

$$g_1 : \sigma - \sigma(max) \leq 0$$
$$g_2 : \delta - \delta(max) \leq 0$$
$$g_3 \; -5mm \leq x_i \leq +5mm$$

x : $sequence$ $\quad$ σ : $stresses$ $\quad$ δ: $deformation$

x_i : $nominal\ weld\ sequence$ $\quad$ g : $contraints$

$y_1(x)$: $analysis\ output\ of\ discipline$ 1

$y_2(x)$: $analysis\ output\ of\ discipline$ 2

$y_3(x)$: $analysis\ output\ of\ discipline$ 3

9.2.1 Sub-system level

The sub-system level is developed based on Individual discipline feasible principles. IDF architecture works on the principle that it solves each discipline independently. The reason for choosing IDF over other architecture is that there are no sensitivities that exist between different disciplines. For example, the result from one discipline i.e. geometrical assurance with respect to the position of spot weld does not have any effect on other disciplines i.e. strength and residual stress analysis during evaluation. Also, IDF architecture results in overall less computational costs than other architecture which then provides the ability to run different disciplines in parallel.

Once the suitable architecture i.e., IDF is chosen, the next step was to formulate the problem. The objective of this sub-system level is to minimize RMS, which is a response from geometrical assurance, Total displacement from strength and von-mises stress from residual stress. The design variable, i.e. the position of a spot weld is the same for all the disciplines. The constraints put forth for this formulation is the search space of 5mm around the nominal weld points.

Once the formulation is completed, the Matlab programming tool was used to evaluate this design problem. Firstly, the search algorithm searches the position of spot weld around nominal weld points and prepares the input sequences for all three disciplines individually. The input sequences refer to a similar change in the position of spot weld with respect to distance and direction to all seven weld points in the test case model. The total number of input sequences prepared for each discipline is 30. During the sub-system level optimization, each discipline will evaluate 30 different sequences of the spot weld. Hence, the total number of iterations is fixed as 90 for the sub-system level. Though the design variable, i.e., the position of a spot weld is the same, the input type for each discipline during evaluation is different. For example, the position of a spot weld is given as WP (weld point) number for geometrical assurance, mesh node number for strength and coordinate data for residual stress. These inputs are connected to respective disciplines for evaluation. As the connection is established already between the software, it starts to evaluate all the disciplines in parallel.

9.2.1.1 Results and discussion

Once all the design variables were evaluated, the next step was to analyze the results. The results from sub-system level optimization is shown in Appendix A and Appendix B. The results will be discussed individually with respect to each discipline.

The RMS results from RD&T with respect to the position of the spot weld is shown in Fig. 48.a. From the graph, it is seen clearly that the spot weld within 2mm and 4mm showed the most significant variations when compared to other positions of the spot weld. Also, there is not much geometrical variation was seen within 1mm. It is also important to discuss the results based on direction from the nominal weld position. Fig. 48.b shows that the variation is seen more when the position is changed below nominal spot weld. Lastly, the minimum variation was seen when the position is changed to right irrespective of any distance from nominal spot weld.

The total displacement results from Ansys with respect to the position of spot the weld is shown in Fig. 49.a In strength analysis also, there are not many changes in strength when the spot weld is moved within 1mm. Also, from Fig. 49.a it is seen clearly that the total displacement of the test case assembly increases when the position of a spot weld is moved away from the nominal weld point. There are some special cases in which even the position of spot weld within 5mm range shows the minimum displacement result. This can be studied by the direction of spot weld with respect to the position of the spot weld. From Fig. 49.b it is evident that the displacement decreases exponentially when the position of spot weld changed to left of nominal irrespective of any distance.

After analyzing all the results, the two optimized sequences from each discipline are independently chosen. These sequences from each discipline were chosen for the next step i.e., system-level optimization. The setup developed to perform system-level optimization will be discussed in the next section.

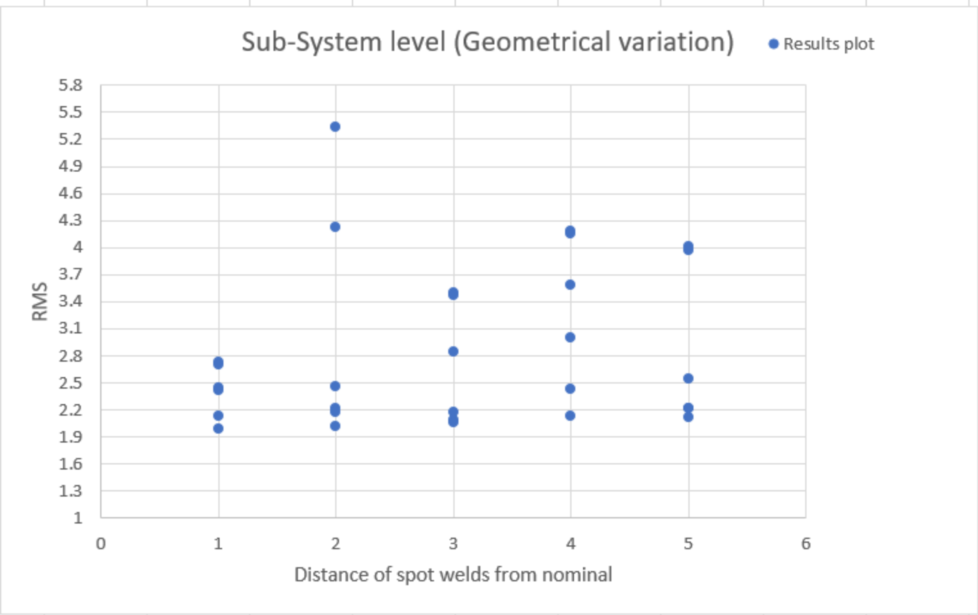


Fig. 48.a: Geometrical assurance Results with respect to the position of spot weld in (mm) based on distance

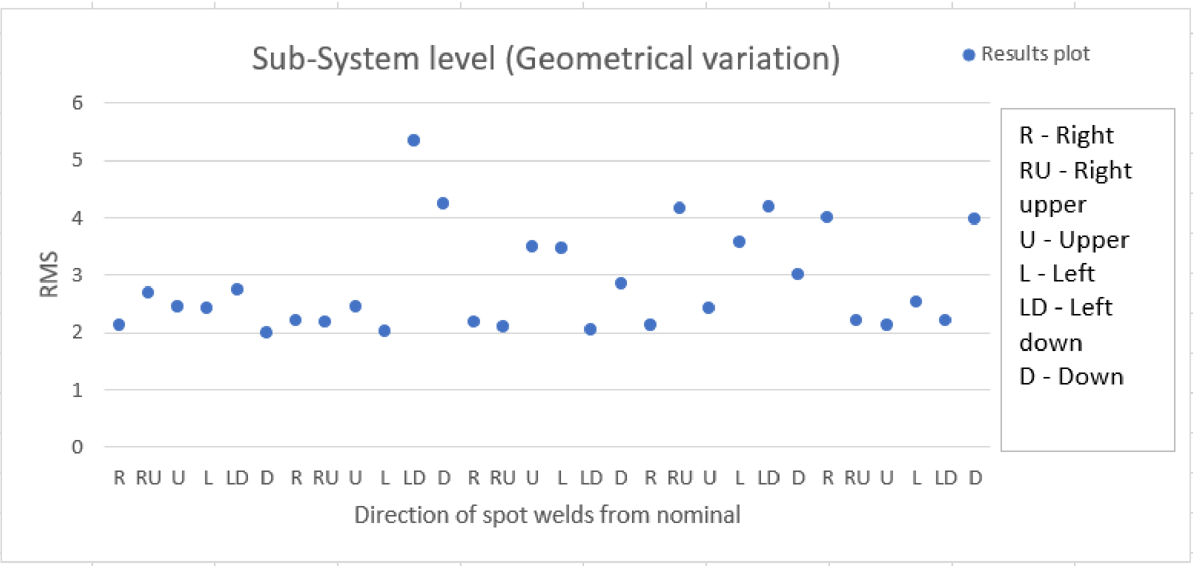


Fig 48.b: Geometrical assurance Results with respect to position of spot weld in (mm) based on direction

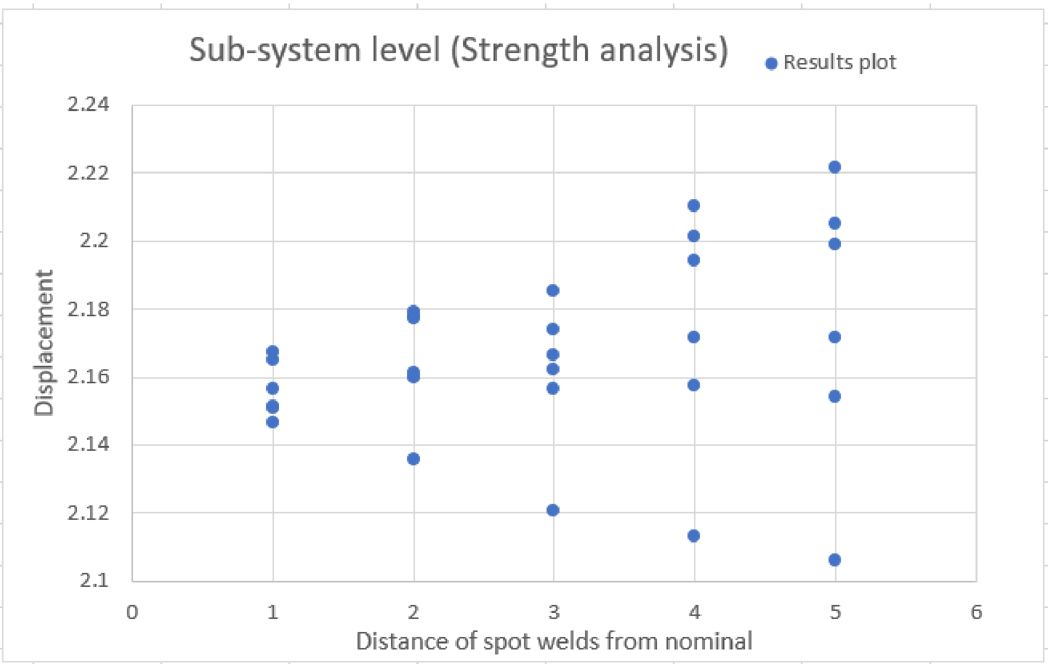


Fig. 49.a: Strength Results with respect to the position of spot weld based on distance.

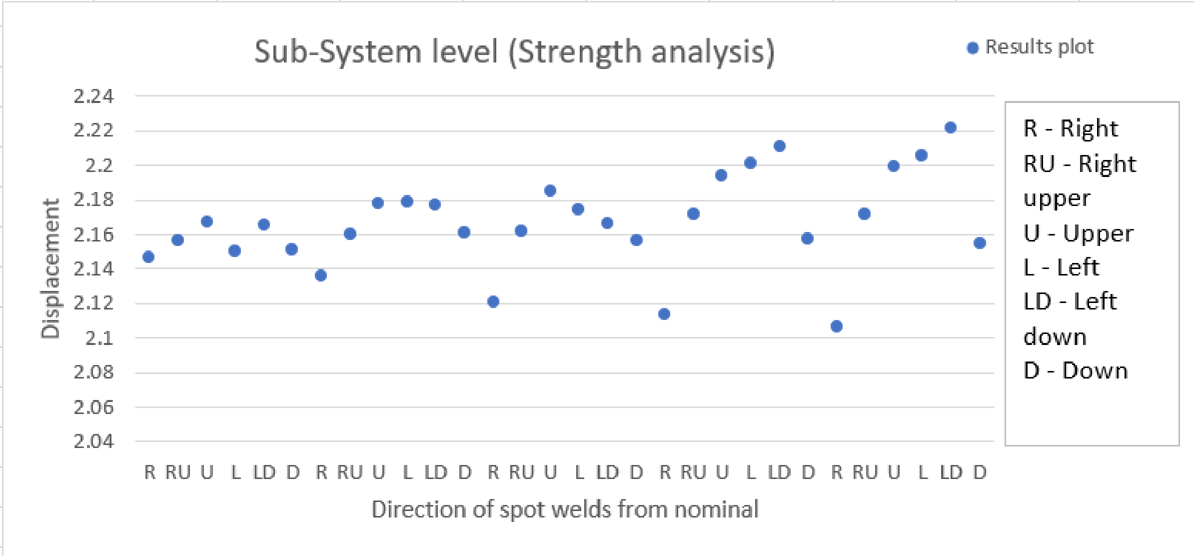


Fig. 49.b: Strength Results with respect to the position of spot weld based on direction

9.2.2 System level

The system level is framed which focuses on minimizing the geometrical variations. From the sub-system level optimization, the two optimized sequences from each discipline were considered as the input sequences for system-level optimization. The objective of this formulation was to minimize RMS. In this formulation, strength and stresses are constrained to obtain sequences with minimum variation. During sub-system level optimization, the position and direction in which the weld points changed are the same with respect to all seven nominal weld points, their combination was not evaluated as the number of sequences will increase and also, in turn, increases the computational time. Since the number of sequences is reduced to six for system-level optimization, the combination of position and direction with respect to all seven nominal weld points was considered. The combination of these two optimized sequences from each discipline was generated with the help of Matlab. From the combination program developed in Matlab as shown in Appendix D, there were 124 sequences generated. These sequences are provided as an input variable for system-level optimization. Hence, the total number of iterations for the system-level is 124.

Once the input sequences are fixed, the next step was to formulate the problem. The objective of this system-level optimization is to minimize RMS, which is a response from geometrical variation discipline. The design variable remains the same i.e., the position of the spot weld. Unlike sub-system level optimization, the strength and residual stress discipline are formulated as constraints in the system level. The constraints from these two disciplines are maximum allowable total deformation and von-mises stress. The maximum allowable values are chosen based on the results from the sub-system level optimization.

Once the system level formulation was framed, the Matlab programming tool was used to evaluate all the sequences generated from the combination of optimized sequence in the sub-system level. Firstly, the sequence was evaluated for strength and residual stress discipline to see if it satisfies the constraints before it is evaluated for geometrical variations. Once the sequences were satisfied by the constraints then the same sequence is provided to evaluate geometrical variation. The process was repeated until all the sequences were evaluated in the same way. The evaluation time required to run system-level optimization is high when compared to sub-system level optimization. There are two reasons for the increase in computational time, firstly, of course, the number of sequences was increased and lastly, unlike subsystem level optimization, the evaluation of sequences in all the disciplines was not done in parallel.

9.2.2.1 Results and discussion

Once all the sequences are evaluated, the next step was to analyze the results. In total, 11 optimized sequences satisfied the objective and constraints mentioned above and constraints. The optimized sequences from system-level optimization are shown in Table 7. Fig. 50 shows the RMS values of optimized sequences and their corresponding total displacement and residual stress values. The trade-off of analysis between different disciplines was performed to suggest three optimized sequences. Form this analysis, the three optimized sequences were chosen namely 1st, 5th, and 11th from Table 5. These optimized sequences are also evaluated against a nominal weld sequence. The comparison of the result can be seen in Table 8. From the comparison, it can be seen that there is a 25% improvement in minimizing the geometrical variation.

S.no	Displacement	RMS	Stress
1	2,1178	2,11	1,39E+09
2	2,1165	2,21	1,51E+09
3	2,1188	2,09	2,03E+09
4	2,2934	2,45	2,04E+09
5	2,1119	2,15	1,05E+09
6	2,1142	2,2	2,03E+09
7	2,117	2,7	1,38E+09
8	2,1157	2,15	2,04E+09
9	2,2896	2,68	2,04E+09
10	2,1179	2,17	2.05E+09
11	2,1124	2,09	1.68E+09

Table 7: Results from system level

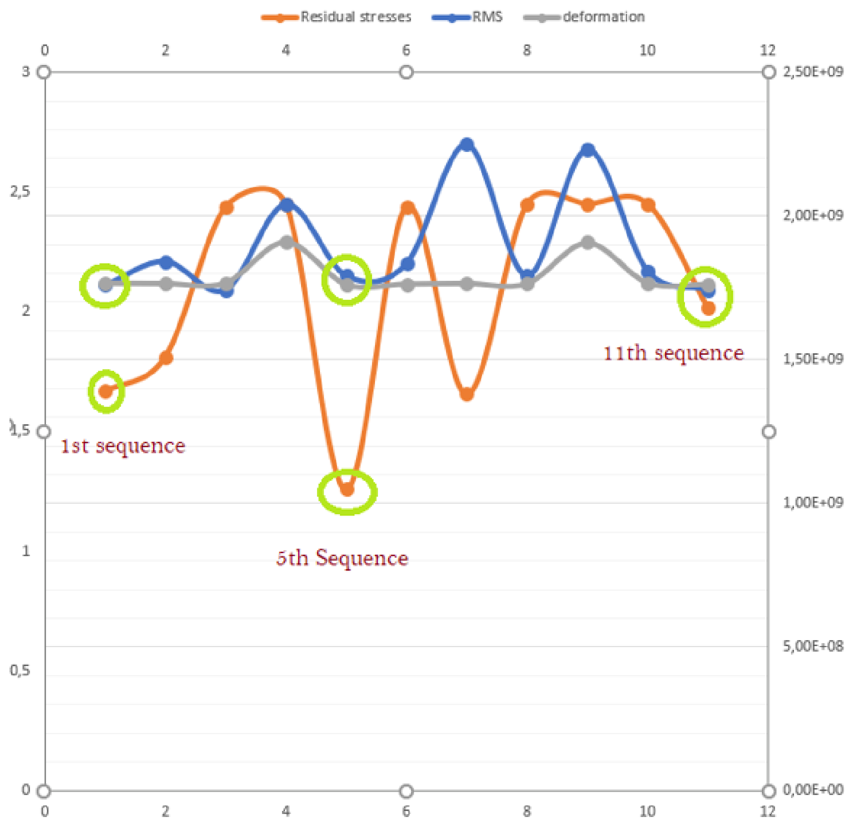


Fig. 50: System-level results plot

S.NO	Spot weld position	RMS	Residual stress	Displacement
1.	Nominal weld position	2,58	2,7e9 pa	2,156 mm
2.	1st optimized position	2,11	1,39 e9 pa	2,1178 mm
3.	2nd optimized position	2,15	1,05 e9 pa	2,119 mm
4.	3nd Optimized position	2,09	1,68 e9 pa	2,114 mm

Table 8: Final optimized results

10 Conclusion

The aim of this thesis was optimizing the position of spot weld with respect to disciplines such as geometrical assurance, strength analysis and residual stress using a multidisciplinary optimization method. From the optimization results, it can be seen clearly each discipline is affected by the change in positions of the spot weld. The results from system-level optimization show that variation between each discipline with respect to the position of a spot weld is different. Hence, a trade-off analysis is conducted keeping in mind that the objective was to minimize geometrical variation and also to satisfy the constraints from strength and residual stress. From the trade-off analysis, it can be seen that there is a significant improvement by 20-25% using the final three sequences compared to the nominal weld sequence.

From the MDO results, it is seen clearly that the position of spot weld has an effect on all three disciplines. The magnitude of the effect on each discipline is heterogeneous. Therefore, it is important to consider the dependency between the position of spot weld and its effect on all the disciplines during the development stage. By choosing the optimal position of spot weld, the effect of geometrical variation in a final assembly can be minimized.

Apart from MDO, a search algorithm was developed to create and initialize the input sequences for each discipline. This method helped in reducing the computational time when compared to traditional search algorithms. Also, a method was developed to establish a connection between different software platforms. This allowed us to evaluate the different position of spot weld with respect to each discipline in a swift way.

The MDO formulation developed for the test case model provided a significant result, which not only shows that the effect of geometrical variation in a final assembly can be minimized but also satisfying the constraints of strength and residual stresses. With this in mind, to use this same method for different sub-assemblies, a little change in the MDO formulation is required. The changes may depend upon the objective of the organization. However, this thesis did not focus on testing the same MDO formulation for different subassemblies.

One of the main objectives of this thesis was to prepare the test case model for evaluation of spot weld position with respect to each discipline. To do this, two different software platforms were chosen i.e. RD&T (Geometrical variation) and Ansys (Strength and Residual stress). Using the above software, the test case model was prepared to analyze the effect of the position of the spot weld. The different procedures mentioned in this thesis to prepare the test case can be used as a platform for modeling different subassemblies with similar objectives.

To conclude, the method used to establish a connection between Ansys and Matlab to perform MDO is demonstrated in this thesis. This method saved a lot of time in evaluating different positions of spot the weld with respect to strength analysis.

11 Recommendations

- This method was developed by only incorporating the disciplines such as geometrical assurance, strength and residual stress but there are other disciplines such as crash analysis which is a dynamic analysis that can be included to increase the robustness of the results. The disadvantage is that the complexity of the simulation increases since larger models need to be simulated.

- The search algorithm was developed during the course of this thesis, and there are room for improvements to make it robust to changes.

- In this thesis, thermal analysis was only conducted based on steady-state thermal and static-structural analysis. A third module which is thermal-electrical can also be introduced, since residual stress of spot weld is caused by the combination of electrical, thermal and structural properties.

Bibliography

[1] Söderberg, R., Wärmefjord, K., Lindkvist, L., & Berlin, R. (2012). The influence of spot weld position variation on geometrical quality. *CIRP Annals*, *volume 61*, 13–16. Retrieved from https://www.sciencedirect.com/science/article/pii/S0007850612001291#!

[2] Söderberg, R., Lindkvist, L., Wärmefjord, K., & Carlson, J. S. (2016). Virtual Geometry Assurance Process and Toolbox. *Procedia CIRP*, *volume 43*, 3–12. Retrieved from https://www.sciencedirect.com/science/article/pii/S221282711600322X#!.

[3] Kristina Wärmefjord, Rikard Söderberg, Björn Lindiu, Lars Lindkvist & Samuel Lorin (2016). *'Joining in Non-rigid variation simulation',* Retrieved from https://www.intechopen.com/books/computer-aided-technologies-applications-in-engineering-and-medicine/joining-in-nonrigid-variation-simulation.

[4] Govik, A. (2014). *Finite Element Analysis of Sheet Metal Assemblies* (dissertation). LiU-Tryck, Linköping, Sweden. Retrieved from http://www.divaportal.org/smash/get/diva2:717630/FULLTEXT02.

[5] Govik, A., Nilsson, L., & Moshfegh, R. (2012). Finite element simulation of the manufacturing process chain of a sheet metal assembly. *Journal of Materials Processing Technology*, *volume 212*(issue 7), 1453–1462. Retrieved from https://www.sciencedirect.com/science/article/pii/S0924013612000635#

[6] Kumar, K. A. (2016, January). Retrieved from https://www.slideshare.net/AjaykumarKalapu/biw-with-definitions.

[7] Roham Sadeghi Tabar, Wärmefjord, K., & Söderberg, R. (2018). Evaluating evolutionary algorithms on spot welding sequence optimization with respect to geometrical variation. *Procedia CIRP*, *Volume 75*, 421–426. Retrieved from https://www.sciencedirect.com/science/article/pii/S2212827118305778

[8] Löveborn, D. (2016). *3D Fe Simulations of Resistance Spot Welding* (dissertation). STOCKHOLM, SWEDEN. Retrieved from http://www.diva-portal.org/smash/get/diva2:915628/FULLTEXT01.pdf.

[9] Klinger , J. F. (2012). *Automotive Body Structure Assembly* (dissertation). Stockholm, Sweden. Retrieved from http://www.diva-portal.org/smash/get/diva2:608422/FULLTEXT01

[10] Roham Sadeghi Tabar, Wärmefjord, K., & Söderberg Rikard Söderberg, R. (2019). A method for identification and sequence optimisation of geometry spot welds in a digital twin context. Retrieved from https://journals.sagepub.com/doi/full/10.1177/0954406219854466.

[11] Söderberg, R., Lindkvist, L., Wärmefjord, K., & Carlson, J. S. (2016). Virtual Geometry Assurance Process and Toolbox. *Procedia CIRP*, *volume 43*, 3–12. Retrieved from https://www.sciencedirect.com/science/article/pii/S221282711600322X#!.

[12] Liu. S. Charles, Hu. S. Jack (1997). 'Variation simulation for deformable sheet metal assemblies using finite element methods'. Retrieved from https://asmedigitalcollection.asme.org/manufacturingscience/article/119/3/368/420822/Variation-Simulation-for-Deformable-Sheet-Metal.

[13] Rihan, Y., Ayyad, S., & Elamy, M. I. (2015). A Study on the Residual Stresses in Resistance Spot Welding. Retrieved from

https://www.researchgate.net/publication/325121721_A_Study_on_the_Residual_Stresses_in_Resistance_Spot_Welding.

[14] Cramer, E. J., Dennis, J. E., Frank, P. D., Lewis, R. M., & Shubin, G. R. (1994). Problem Formulation for Multidisciplinary Optimization. Retrieved from https://pdfs.semanticscholar.org/5570/ceb148ddbcc54e959c1d873416a282e78ccc.pdf

[15] Cramer, E. J., Frank, P. D., Shubin, G. R., Dennis, J. E., & Lewis, R. M. (2012). On Alternative Problem Formulations for Multidisciplinary Design Optimization. Retrieved from https://pdfs.semanticscholar.org/a27e/507fa65e9084f8dd7188e9f4881bb057698c.pdf.

[16] Kachitvichyanukul, V. (2012). Comparison of Three Evolutionary Algorithms: GA, PSO, and DE. Retrieved from https://www.researchgate.net/publication/260393066_Comparison_of_three_evolutionary_algorithms_GA_PSO_and_DE.

[17] José Alexandre Tavares Guerreiro Fregnani, & Mattos, B. S. de. (2018). An Example of Genetic Algorithms Applied to Multidisciplinary Design Optimization: The 3D Payload-Range-DOC Envelope. Retrieved from https://www.researchgate.net/publication/324121716_AN_EXAMPLE_OF_GENETIC_ALGORITHMS_APPLIED_TO_MULTIDISCIPLINARY_DESIGN_OPTIMIZATION_THE_3D_PAYLOAD-RANGE-DOC_ENVELOPE.

[18] Dépincé, P., Guédas, B., & Picard, J. (2010). Multidisciplinary and multiobjective optimization: Comparison of several methods. Retrieved from https://hal.archives-ouvertes.fr/hal-00449605/document.

[19] Sokolowski, J. A. (1953). *Principles of modelling and simulation*. Retrieved from https://books.google.se/books?hl=en&lr=&id=QrTJiwTwSG8C&oi=fnd&pg=PT8&dq=john a Sokolowski &ots=WKBHTZxAq3&sig=YfXvqiUyHePvtoCxy8d9ArBWj_g&redir_esc=y#v=onepage&q=john a Sokolowski&f=false

[20] Banks, J. (1998). Handbook of Simulation Principles, Methodology, Advances, Applications, and Practice.

[21] Spot welding technical. Retrieved from https://www.pwrwp.co.uk/spot-welding-technical/

[22] SPOT WELDING TECHNICAL INFORMATION. Retrieved from https://www.titespot.com/spot-welding-technical-information/

Appendix A – Geometrical variation

8	38	68	97	122	152	182	2.13
9	39	69	98	123	153	183	2.7
10	40	70	99	124	154	184	2.44
11	41	71	100	125	155	185	2.42
12	42	72	101	126	156	186	2.73
13	43	73	0	127	157	187	1.988
2mm							
14	44	74	102	128	158	188	2.22
18	48	78	106	132	162	192	2.17
22	52	82	110	136	166	196	2.46
26	56	86	114	140	170	200	2.014
30	60	90	118	144	174	204	5.33
34	64	93	0	148	178	208	4.23
3mm							
15	45	75	103	129	159	189	2.17
19	49	79	107	133	163	193	2.09
23	53	83	111	137	167	197	3.5
27	57	87	115	141	171	201	3.47
31	61	91	119	145	175	205	2.06
35	65	94	0	149	179	209	2.84
4mm							
16	46	76	104	130	160	190	2.13
20	50	80	108	134	164	194	4.16
24	54	84	112	138	168	198	2.43
28	58	88	116	142	172	202	3.58
32	62	92	120	146	176	206	4.18
36	66	95	0	150	180	210	3.001
5mm							
17	47	77	105	131	161	191	4.004
21	51	81	109	135	165	195	2.22
25	55	85	113	139	169	199	2.118
29	59	89	117	143	173	203	2.54
33	63	0	121	147	177	207	2.21
37	67	96	0	151	181	211	3.97

Appendix B – Strength analysis

10763	15464	14066	11179	15250	9247	10138	2.1467mm
7236	15460	14088	7046	15275	9237	10102	2.1563
10973	15467	14076	11478	15282	9101	995	2.1675
11095	15530	14123	11346	15258	9276	9970	2.1506
7237	15501	14090	7047	15206	9179	10038	2.1651
10811	15538	14171	11151	15231	9144	10039	2.1514
2mm							
10696	15419	14204	11223	15180	9198	9889	2.1357
7236	15403	14007	7045	15165	9095	9879	2.1598
11035	15414	14037	11392	15198	9233	9827	2.1775
10924	15425	14035	11364	15183	9263	9828	2.1790
7238	15413	14010	7048	15327	9091	10087	2.1772
10886	15443	14036	11288	15171	9104	10094	2.1612
3mm							
10757	15397	14119	11186	15142	9121	10089	2.1207
7235	15473	14089	7044	15284	9080	10133	2.1621
10930	15465	14059	11327	7407	9082	9912	2.1851
11015	7372	13999	11434	7410	9055	1074	0.72931
7239	15488	14211	7031	15380	9176	9956	2.1663
10881	15629	14002	11147	15145	9154	9874	2.1567
4mm							
10723	15434	14051	11298	15191	9220	9990	2.1132
7234	15608	7756	7043	15151	9137	9834	2.1718
11064	7369	14207	11363	15963	9238	10042	2.1943
10909	9708	14040	11488	16085	9119	9994	2.2012
7240	7376	14132	7183	15168	9096	9971	2.2104
10882	15399	14042	7222	15334	9092	10031	2.1574
5mm							
10704	15516	14160	11301	15241	9157	9818	2.1062
7233	15395	14752	7042	15340	9254	10058	2.1717
10950	9727	7761	11437	16030	9153	7011	2.1988
11059	9656	14185	7052	16015	9271	9856	2.2053
7241	9740	14033	7184	15365	9162	9906	2.2215
10679	15635	14111	9674	15358	9136	9982	2.1542

Appendix C - Search_algorithm

```
clear all;

clc;

clearvars;

fid = fopen('tabledata5.txt', 'w');

fclose(fid);

data1 = xlsread('data_sorting.xlsx');

node_number = data1(:,1);

x = data1(:,2); % column 2 is assigned the variable x

y = data1(:,3); % column 3 is assigned the variable y

z = data1(:,4); % column 4 is assigned the variable z

data2 = [x y z];


%%  Nominal Weld position from the test case spot weld

Seq = [4313.07 -462.32  705.54;4288.92 -475.86  684.92; 4277.09 -458.22  650.28;4289.41 -
518.11  676.55;4289.92 -545.38  704.72;4283.62 -580.72  681.7;4285.78 -577.43  710.45];

for i = 1:7

 A = Seq(i,:)

%% for loop for finding the distance between two values

 for i = 1:length(data2)

 n(i) = norm (A-data2(i,:));

end

n_4 =transpose(n);

n_3 = [node_number];


%% node number within 1mm

idx = find(n_4>=0.5 & n_4<=1.5);

node_1 = [n_3(idx),data2(idx,:),n_4(idx)];


%% Node number within 2mm

idx_1 = find(n_4>=1.5 & n_4<=2.5);
```

```
node_2 = [n_3(idx_1),data2(idx_1,:),n_4(idx_1)];

%% Node number within 3mm
idx_2 = find(n_4>=2.5 & n_4<=3.5);
node_3 = [n_3(idx_2),data2(idx_2,:),n_4(idx_2)];

%% node number within 4mm
idx_3 = find(n_4>=5 & n_4<=6);
node_4 = [n_3(idx_3),data2(idx_3,:),n_4(idx_3)];

% node number within 5mm
idx_4 = find(n_4>=4.5 & n_4<=5);
node_5 = [n_3(idx_4),data2(idx_4,:),n_4(idx_4)];

%% all node numbers
Spotweld_1 = [node_1; node_2; node_3; node_4; node_5];%insert node 6 if required
Table_1 = table(Spotweld_1);
M = table2array(Table_1);
dlmwrite('tabledata5.txt', M, '-append','delimiter',' ', 'newline', 'pc');
end
```

Search algorithm code developed in Matlab

Appendix D - Combination program

```
A = [847 10723];
B= [15538 15434];
C=[14171 14051];
D= [1151 11298];
E=[15231 15191];
F=[9144 9200];
G=[10039 9990];

k = combvec(A,B,C,D,E,F,G)
```

Combination program developed in Matlab