FEM-based Virtual Manufacturing and Testing Procedure for Resistance Spot Welding

M. N. Yusuf¹, W. E. W. Rahman¹ and Y. H. P. Manurung*¹
¹Faculty of Mechanical Engineering, UiTM Shah Alam, Selangor, Malaysia

*yupiter.manurung@salam.edu.my

Abstract. This research focused on the development of numerical approximation procedure applied for resistance spot welding process which includes testing procedure to evaluate the tensile strength of a spot welded joint. The spot weld is modelled according to standardized dimension for tensile test with main material properties of Cu-sw as electrode and low carbon steel S235 as sheets with 1mm thickness. It is including electric conductivity, resistivity and heat transfer coefficient in solid body as well as contact interface. The FEM simulation is conducted using the process parameter of current 15,000A, force 5,000 N and different stages of time following the welding process and tensile test which is simulated after released both electrodes and material reaches the initial temperature with contact clamp velocity of 5mm/min. It is expected that the resistance spot welding process can be successfully simulated and producing result within acceptable range of errors.

1. Introduction
Resistance spot welding (RSW) known as one of the most common techniques applied for joining metal sheets. This metal joining techniques is broadly used in manufacturing industry such as automotive which vehicle body and parts are assembled. This welding technique was preferred by the manufacturers for metal sheet related products due to the advantage of time and cost efficient [1]. Among of the important parameters required in resistance spot welding process are heat, pressure and time [2]. The fusion of resistance spot welded joint is represented by weld nugget which is formed as a result of heat generated due to resistance to the high current flow at low voltage [2]. The quality of spot welded joint is highly influenced by the welding current, current flow time, welding tips contact surface, cross-sectional area and sheet thickness.

In the present study, Finite Element Method (FEM) is used to model and simulate the resistance spot welding process to investigate the thermal, electrical and mechanical behaviour of a spot welded joint. Finite Element Method (FEM) was known as one of efficient methods to analyse different phenomenon of engineering problem as it has the capability to produce a much more detailed set of results than experimental investigations.

In spot welding, suitable process parameter is crucial to be used on a specific set of material properties to achieve a desired and acceptable result. For example, size of weld nugget and electrode indentation are depending on the electrode force, weld current and weld time. These parameters are also crucial to achieve high strength of spot welded joint [3]. It was one of the reasons that many researchers were focused their studies on parameter optimization in resistance spot welding.
Resistance spot welding physics is governed according to Ohm’s law, where I represents the current, E is voltage drop across the electrodes and R is the resistance through the material in ohms. R represents the total resistances in the form of contact and material resistance. Therefore, for a given value of R, the magnitude of I is determined by E. The transformer controlled the current to convert from alternating current to a low voltage which then delivered to a weld of a given resistance [4]. The resistance spot welding process is defined using the following mathematic equation:

\[ E = IR \]  

(1)

The total heat (H) generated in watt-seconds at the spot welded joint, can be expressed using the formula as follow:

\[ H = PRT \text{ or } H = IET \]

\[ T \text{ is time in seconds.} \]  

(2)

Figure 1 below shows type of resistance which exist during the spot welding process:

![Figure 1. Example model of resistance spot welding process](image)

2. Virtual Manufacturing Simulation of Resistance Spot Welding process using MSC. Marc.

MSC. Marc is an advanced non-linear finite element software which capable to analyze and simulate product behavior under static, dynamic and multi-physics loading condition [5]. MSC. Marc versatility in modeling nonlinear material behaviors and transient environmental conditions makes it ideal to solve different kind of complex design and engineering problems. Founded since 1971, the software was widely used in several industries such as automotive, aerospace, machinery, electronics, biomedical, oil and gas etc.

For resistance spot welding, the simulation in MSC. Marc involved 3 stages; pre-processing, solving and post-processing. The process is illustrated in figure 2 below.

![Figure 2. Resistance spot welding procedure in MSC. Marc](image)

The component involved for spot welding modelling in MSC. Marc are two metal sheets overlap each other. The length of the specimen is 100mm along the x-axis while the width is 50mm along the z-axis with thickness of 1mm each. Symmetrical electrode (truncated shape) is drawn with the tip surface of the electrode facing the outer surface of the sheets. Dimension of the electrode used in simulation is
15mm length along the y-axis with 16mm body diameter and 4mm tip diameter. Figure 3 shows the modelling of resistance spot welding in MSC. Marc:

![Figure 3. Details modelling of resistance spot welding in MSC. Marc](image)

Tetrahedral-mesh is applied to the electrodes to accommodate the truncated shape while hexahedron-mesh is applied to the specimen. For resistance spot welding process, analysis class required in the simulation is current, thermal & structural.

To accommodate the process parameter of resistance spot welding in MARC, parameter table & coordinate system were created due to most of the properties used are temperature dependent and not constant along the processes. Independent variables which is required to be defined in the simulation are current, electrode force, thermal contact heat transfer coefficient, electricity contact conductivity, young modulus, thermal expansion coefficient, thermal conductivity, specific heat, electric resistivity, electric conductivity.

In this simulation material used for the specimen is S235-JMP-MPM-sw low carbon steel while Cu-Sw copper is used for both of the electrodes. Table 1 shows the S235 chemical composition while table 2 indicates the main parameter setting used in simulating the resistance spot welding process.

| Parameter       | Value |
|-----------------|-------|
| Force (kN)      | 5     |
| Current (kA)    | 15    |
| Hold time (cycle)| 2    |
| Weld time (cycle) | 15   |
| Squeeze time (cycle) | 2    |

The plasticity data used for the metal sheet material is based on S235-JMP-MPM-sw flow curve which was created and converted into .mat file. Figure 4 shows the flow curve for S235 material.
Contact body and interaction are another key parameter to be defined when simulating resistance spot welding process in MSC. Marc especially with RSW process involved analysis of electrical, thermal & structural. The heat generated to form a fusion was because of resistance to the high value of current flow in the form of contact and material resistance. Other than resistance which exist in the material, contact resistance/conductance for both electrical & thermal are vital in RSW simulation. The relationship was defined through contact table option.

Before the simulation of RSW process can be run, the relevant boundary conditions are to be defined in MSC. Marc. The boundary input forms all physical instruction to the welding process. Among of the boundary condition required to be set for RSW are boundary on positioning of the specimen and movement of the electrode, point load for the electrode pressure at 5kN, value of the weld current at 15kA, fix temperature after cooling, tensile test displacement at constant speed of 5mm/min.

Prior to running the simulation, load cases will be created, and the job parameter will be defined. 3 load cases were created each for welding, cooling and tensile test. For S235 material, user subroutine UACT GLUE will activate the glue function at the node of specimen that reach welding temperature of approximately 1,500-1600 °C. This will define the fusion area for the spot welding at the end of the simulation process.

3. Results and Discussion
Figure 5 shows the contact condition between each of the components, top electrode, top sheet, bottom sheet and bottom electrode. The contact status at the end of the simulation indicates that the contact is exist between each of the components which is crucial for contact resistance to generate the heat at the spot-welded point.

Electric potential result as shown in figure 6 indicates the current flow through the top electrode, specimen and bottom electrode. The point current was applied to the top surface of the top electrode while zero voltage is set at the bottom surface of the bottom electrode.

Figure 7 shows the formation of weld nugget at temperature >1500°C during the resistance spot welding process. The peak temperature is also used to define the fusion area with the application of UACT GLUE user subroutine. All nodes at the contact surface between the two sheets will be subjected to glue deactivation condition except those nodes that reach the pre-defined temperature in the user subroutine.

Figure 8 shows the sheets which were subjected to tensile shear force at the end of the simulation. High stress region is represented with yellow and orange region when the contact clamp velocity of 5mm/min is applied.
Figure 5. Contact Status

Figure 6. Electric Potential

Figure 7. Formation of weld nugget
For resistance spot welding process simulation in MSC. Marc, process parameter was recognized as key to obtain successful result for welding simulation and tensile test. After the simulation, it was found that welding current \( (I) \) was known to have highest impact on the simulation result while each of the process welding time had also play significant factor to the result.

Other influential factor that was recognized after the simulation is the electrical and thermal contact conductivity or resistance. In most cases the electrical contact resistance is 30 time larger than the electrical material resistance. Electrical material resistance and electrical contact resistance are strongly depending on temperature. Additionally, the electrical contact resistance is also depending on the contact pressure. During the welding process the electrical contact resistance decreases rapidly whereas the electrical material resistance increases. Figure 9 shows the time curves of electrical material resistance, electrical contact resistance and the complete resistance of the system during the spot welding process.

**Figure 8.** Tensile modelling in MSC Marc

**Figure 9.** Time curves for electrical resistances during a resistance welding process

### 4. Conclusion

Through Finite Element Method (FEM), resistance spot welding process with tensile test had been modelled. Result produced from the simulation highlights several critical findings as follows:

1. The resistance spot welding process was successfully simulated using MSC. Marc mentat software with suitable materials and process parameters were defined in clear and structured procedure to achieve the expected and desire result.

2. Other than material properties and boundary condition setting, key parameters such as weld current, electrode pressure, and welding time is found to be significance and even small changes in the value, resulting to simulation error and in some cases consume longer simulation time.

The result data obtain through the FEM simulation is useful to establish deeper understanding on the resistance spot welding process and behaviour. Information collected in the simulation is also useful to minimize the gap between simulation and experimental result. The procedures implemented in the simulation will be further developed to explore more knowledge in resistance spot welding.
behaviour such as effect of parameter change to fatigue strength and use of suitable damage model to investigate and characterize the strength of the spot welded joint under different load conditions. This research will contribute towards further development of research in Virtual Manufacturing by considering the material behaviour and modelling.

Acknowledgement
The collaborating authors would like to express highly gratitude to the associates in Advanced Manufacturing Technology Excellence Centre (AMTEx), Faculty of Mechanical Engineering, Universiti Teknologi MARA (UiTM) in Malaysia for their contribution in this research.

References
[1] Joel Andersson, Jonatan Deleskog, Fatigue Life and Stiffness of the Spider Spot Weld Model, Master’s thesis, ISSN 1652-8557, 2014:22
[2] Ario Sunar Baskoro, M. Rizky Trianda, Jos Istiyanto, Sugeng Supriyadi, Danardono A. Sumarsono, Gandjar Kiswanto, Effects of Welding Time and Welding Current to Weld Nugget and Shear Load on Electrical Resistance Spot Welding of Cold Rolled Sheet for Body Construction, 2014 IEEE International Conference.
[3] Manoj Raut and Vishal Achwal, Optimization Of Spot Welding Process Parameters for Maximum Tensile Strength, International Journal of Mechanical Engineering and Robotic Research, Vol. 3, No. 4, October 2014
[4] Miller Technical Handbook for Resistance Spot Welding, 2012-003 335D, Miller Electric Mfg. Co.
[5] Qilei Sun, Zerui Liu, Long Yang, Application of Finite Element Analysis Software MSC.Marc in Material Tests, Advanced Materials Research, Vols. 945-949, pp 95-98, 2014