Finite element analysis of automotive arm brake pedal for rapid manufacturing

MIM Sargini1,3, SH Masood2, Suresh Palanisamy2,*, Elammaran Jayamani1 and Ajay Kapoor2

1Faculty of Engineering, Computing and Science, Swinburne University of Technology (Sarawak), Malaysia
2Dept. of Mechanical and Product Design Engineering, Swinburne University of Technology, Hawthorn, VIC, Australia
3Malaysia Automotive, Robotics and IoT Institute, Cyberjaya, Malaysia

*E-mail: spalanisamy@swin.edu.au

Abstract. Current automotive arm brake pedal uses conventional manufacturing method such as metal press and CO₂ welding processes, which are known to be heavy, costly and demand higher quality control. This paper will discuss how finite element analysis can be applied to optimise the design of a brake pedal for Rapid Manufacturing application based on current additive manufacturing process to produce automotive brake pedals with reduced material weight. The performance of the new design is compared with the current design of arm brake pedal to realise the potential of rapid manufacturing to save time and cost in automotive parts.

1. Introduction
There are many components that control the movement and dynamics of a vehicle. However, there are not many components that are in direct contact with the driver and one of thesekey components is a pedal brake. A pedal brake act as a switch for the driver to stop the vehicle and it is a critical part of the vehicle. The driver pedals the pedal brake to commence the car to stop. The pedal force is then transferred to the hydraulics, which then triggers the braking sequence. During the emergency braking, a huge force from the driver is transferred into the brake system within the least time, making the pedal brake to provide great mechanical strength and resistance to hold the force. Automotive engineers are continually exploring new manufacturing technologies like Additive Manufacturing to reduce the cost of designing and manufacturing key automotive components like brake pedals.

According to ISO/ASTM 52900:2015 [1], Additive Manufacturing (AM) is the process of joining materials to make objects directly from 3D model data, usually layer upon layer, as opposed to subtractive manufacturing methodologies. Synonyms are additive fabrication, additive techniques, additive layer manufacturing, layered manufacturing and solid freeform fabrication [2]. Rapid Manufacturing (RM) is a mass manufacturing technique based on AM technologies. The future of RM is very promising since its demand among the manufacturers continues to expand. We have yet to see the dream of manufacturing engineers today to extend the total automation to functional and full-life components [3] via RM.

RM technologies are creating a world of possibilities that can lead an organization in new directions and help launch new businesses and business models. Wohler’s Report [4] has highlighted
that the final part production has been the highest growing application of RM in recent years and would be the largest and most significant application of RM technology. RM also is used to produce precision parts and final parts for automotive, aerospace, medical and healthcare, and consumer durables industries [5]. In other projects, the researchers believe that as for real direct RM of products, its application is still in its infancy, even though very promising. The real breakthrough of RM will mainly depend on cost and productivity improvements, which have to be accompanied by further technical progress in material properties and most of all in accuracy and reliability [6].

In the automotive industry, Prada [7] successfully applied RM in improving an accelerator pedal for his Formula Student racing car. He came up with two prototypes of an accelerator pedal, both made by using RM technologies which are Fullcore 720 with poly-jet technology, and other prototype uses ABS-plus (P430) with FDM technology. The newly design accelerator pedal resulted in 38% of weight reduction compared to its original design made by 7075 aluminium.

In this paper, a new optimised design of an arm brake pedal, which can be produced by RM technologies, material and process, will be introduced using a commercial finite element analysis (FEA) software. The new design will be evaluated to ensure it reliability for the market. In this study, the current design of an arm brake pedal, which uses conventional manufacturing technologies will also be analysed by FEA to compare with the performance of the new design. Several structural analysis tests will be conducted for both arm brake pedal design to study its performance. Both results will be discussed and compared to verify whether the new design is good enough for the market or not.

2. FEA of arm brake pedal: current design
For this study, the arm brake pedal of a five-door hatchback car will be investigated. Figure 1 shows the computer aided design (CAD) model of the brake pedal. The arm brake pedal generally consists of four child parts, which are boss (1), arm (2), pad (3) and stopper (4) as numbered in figure 1. The boss is made of cold drawn steel and acts as a connection between the arm brake pedal and its mounting. The arm is the main body of the arm brake pedal, made of hot rolled steel, which holds and transfer the braking force accordingly. The pad, made of hot rolled steel, is where the driver rests his/her foot to transfer the braking force during the braking event. The stopper function is to maintain the arm brake pedal location during the non-braking event, thus this child part will be ignored in this study since it is not involved during the braking event. Table 1 shows the mechanical properties of the three main child parts used in the analysis. All three child parts then joined by a CO₂ welding process at two different areas which are boss and arm, and arm with the pad.

![Figure 1. The CAD data of the arm brake pedal.](image)

| #  | Child Parts | Thickness mm | Young Modulus (E), MPa | Poisson’s Ratio, v | Tensile Strength (TS), MPa | Yield Strength (YS), MPa | Density (ρ), Ton/m³ | Manufacturin g Process |
|----|-------------|--------------|------------------------|-------------------|---------------------------|------------------------|---------------------|----------------------|
| 1  | Boss        | 15.1 OD 19.0 | 200,000                | 0.290             | 290                       | 290                    | 7.87                | Metal extrusion       |
| 2  | Arm         | 7.0          | 201,000                | 0.306             | 440                       | 295                    | 7.87                | Metal stamping        |
| 3  | Pad         | 2.3          | 201,000                | 0.306             | 440                       | 295                    | 7.87                | Metal stamping        |
Basically, the arm brake pedal is a simple machine of a lever with a Mechanical Advantage (MA). Figure 2 below shows the arm brake pedal modelled as a simple machine; a lever with MA. Mathematically, we can model the MA to be:

\[
MA = \frac{F_p}{L_1} = \frac{F_d}{L_2}
\]  

(1)

The brake force requirement to test the performance of the arm brake pedal has been determined based on Limpert’s study [8], which suggested that the maximum force exerted with right foot for man is 823N and women is 445N [8]. For maximum performance evaluation, the maximum force of 823N is used in this study. To evaluate the performance of the design, the Finite Element Analysis (FEA) software Altair HyperWorks has been used in this study. 2D meshing (PSHELL) mixed of quad and tria elements have been used with a target elements size of 1mm. For the mechanical connection (CO₂ welding), a 1-D rigid element has been used to represent the mechanical connection. This is to ensure no loss of force transfer during the analysis.

![Figure 2](image1.png)

**Figure 2.** A model of a lever with MA, image courtesy of [8].

Figure 3 shows the boundaries conditions for the arm brake pedal model. The load is applied on one end of the arm. The load will be nodally distributed to the area of pad and arm join. The model is constraint at two areas – the boss and clevis hole of the arm. Rotational of Y-axis is the only degree of freedom enabled for both constraints.

![Figure 3](image2.png)

**Figure 3.** The Boundaries Condition of the arm brake pedal FE model.

To evaluate the performance of the current design of the arm brake pedal, three common structural analysis has been performed in the Altair HyperWorks environment, which includes linear static analysis, non-linear static analysis and fatigue analysis. The linear static analysis will be used to determine the maximum stress the arm brake pedal absorb during the analysis. The non-linear static analysis will be used to determine the maximum deflection/displacement of the subject under the stress. For this study, the geometric non-linearity parameter will be enabled. For evaluation, any displacement/deflection more than 10mm will be considered as a failure. The fatigue test will be used to evaluate the design durability for 1 million test cycle. All the parameters for these tests were set to the default value as suggested by the software developer.

Figure 4 shows the results of the linear static analysis for the current design of the arm brake pedal. Since the arm and the boss use different type of steel materials, they have different limits. For the arm, there is a maximum stress concentration valued at 375.59 MPa located at the clevis hole area while for the boss, maximum stress concentration valued at 174.805 MPa is located at the welding connection.
area. The maximum displacement for the current design of an arm brake pedal is 1.482 mm at the one end of the arm. Table 2 summarises the overall linear analysis result.

**Table 2.** The overall result of linear static analysis.

| Child parts       | Tensile Strength, MPa | Maximum element stress, MPa | Displacement, mm | Safety factor |
|-------------------|------------------------|-----------------------------|------------------|---------------|
| Boss              | 290.000                | 174.805                     |                  | 0.60          |
| Arm               | 440.000                | 375.590                     |                  | 0.85          |
| Arm brake pedal   |                        |                             | 1.482            | 0.85          |

**Figure 4.** The overall result of the linear static analysis; stress concentration of arm (left) and boss (right).

Figure 5 shows the results of non-linear static analysis of the current design. For the arm, there is a maximum stress concentration valued at 294.497 MPa located at the clevis hole area while for the boss, maximum stress concentration valued at 175.950 MPa located at the welding connection area. The maximum stress concentration for non-linear static analysis for the arm is lower than linear static analysis. This is true since, for the non-linear static analysis, the geometric non-linearity property of the material was taken into account. The maximum displacement for the current design of the arm brake pedal is 1.482 mm located at the one end of the arm. Table 3 summarizes the overall non-linear analysis results.

**Figure 5.** The overall result of the non-linear static analysis; stress concentration of arm (left) and boss (right).

Figure 6 shows the result of the fatigue analysis towards the arm brake pedal. This result shows that the design is reliable even after $10^{20}$ cycles. As a conclusion, the current design of the arm brake pedal passes all three structural analysis.
Table 3. The overall result of non-linear static analysis.

| Child parts         | Tensile Strength, MPa | Maximum element stress, MPa | Displacement, mm | Safety factor |
|---------------------|------------------------|-----------------------------|------------------|--------------|
| Boss                | 290.000                | 175.950                     |                  | 0.60         |
| Arm                 | 440.000                | 294.497                     |                  | 0.67         |
| Arm brake pedal     |                        |                             | 1.484            | 0.85         |

Figure 6. Fatigue analysis result.

3. FEA of arm brake pedal: optimized design for RM

Altair Inspire software has been used to come up with a new optimized design of the arm brake pedal that can be manufactured by the rapid manufacturing (RM) process using an appropriate RM material matching the properties of the conventional pedal material. The RM material selected for this study is a metal-polymer composite filament, Ultrafuse 316L by BASF, which is used by the Fused Filament Fabrication (FFF) technology, also known as Fused Deposition Modelling (FDM). Table 4 shows the mechanical properties of Ultrafuse 316L used in the analysis [9].

Table 4. The mechanical properties of Ultrafuse 316L [9].

| Material            | Young Modulus (E), MPa | Tensile Strength (TS), MPa | Yield Strength (YS), MPa | Density (ρ), Ton/m³ | Percentage of Elongation, % |
|---------------------|------------------------|-----------------------------|--------------------------|---------------------|-----------------------------|
| Ultrafuse 316L (FDM)| 152,000                | 465                         | 167                      | 7.88                | 31                          |

A design space with the same boundary conditions has been used as the input for the Altair Inspire, which is then processed to generate a topologically optimized model, eliminating the non-necessary mass in the part, hence reducing the mass of the arm brake pedal without affecting its structural integrity. Figure 7 shows the input data with boundary conditions for the Altair Inspire environment.

After several iterations, Altair Inspire comes up with an organic optimized design of the arm brake pedal. Figure 8 shows the final result of the topology optimized design of the arm brake pedal. The new topology optimized design will be analyzed and compared to the performance of the current arm brake pedal. All three structural analyses have been performed with this new design in the Altair HyperWorks environment, identically as applied with the current design.

Figure 9 shows the results of the linear static analysis of the optimized design. Identical to current design results, it is noted that the maximum stress concentration is located at clevis hole area, with a value of 410.830 MPa. The maximum deflection of the new arm brake pedal is 1.76 mm, located at the one end of the pad.

Figure 10 shows the results of the non-linear static analysis for the optimized design. For the non-linear static analysis, the results are also found to be identical with the current design results. The maximum stress concentration is located at clevis hole area, value at 168.943 MPa. The geometric non-linearity property gives an identical result for both designs and the maximum stress is lower. The maximum displacement is found to be 1.84 mm located at one end of the pad.
Figure 7. Input data with identical boundaries conditions.

Figure 8. The topology optimized design of the arm brake pedal, utilizing selected RM material.

Figure 9. The result of linear static analysis for stress concentration (left) and displacement (right).

Figure 10. The result of non-linear static analysis for stress concentration (left) and displacement (right).

Figure 11 shows the fatigue analysis result of the new optimized design. Interestingly, it is noted that the fatigue analysis result for both designs are not identical. The result for new design shows the
variation of the damage result, instead of current design, where the damage result is equal within the pedal brake. This shows that the mass of the new arm brake design has been topology optimized; maximizing the most important area without scarring the structural integrity. The minimum damage fatigue life for the new arm brake design is $1.27 \times 10^6$ cycles located at clevis hole area. The minimum limit for this test is $1.0 \times 10^6$ cycles, hence the new arm brake pedal is reliable and passes the fatigue test. Thus, as a conclusion, the new arm brake design has passed all three structural integrity tests.

![Figure 11. The result of fatigue analysis.](image)

Table 5 summaries the overall performance for the new arm brake pedal design. The result shows a marked reduction of 54% in overall weight of the new optimized design compared to the current design.

| Analysis          | Limit  | Result    | Safety Factor |
|-------------------|--------|-----------|---------------|
| Linear static     | Stress | <465 MPa  | 411 MPa       | 0.88          |
|                   | Displacement | < 10mm | 1.76 mm       | 0.18          |
| Non-linear static | Stress | <465 MPa  | 169 MPa       | 0.36          |
|                   | Displacement | < 10mm | 1.84 mm       | 0.18          |
| Fatigue           | > $10^6$ | $1.27 \times 10^6$ | 1.27 |
| Weight            | 1 Kg   | 0.46 Kg   | 54%           |

4. Conclusions
The results of this study have shown that the current design of the arm brake pedal, manufactured by conventional techniques, has passed all three structural analysis integrity tests, since this design is already in the market. The new optimized design of the arm brake pedal, based on RM material, also passes all three structural integrity tests. This shows that the new pedal design, which uses the RM material and technologies, is also capable to be used for the market. The new design was not only success in passing all three structural integrity test but it also came out to be 54% less in weight.

Based on the result, we can observe that the new arm brake pedal design is applicable to Rapid Manufacturing technologies and can be considered to replace current design based on costly conventional manufacturing technology. The structural analysis results proved that the RM technologies can be applied for the mass market application. A significant weight reduction over the current design without affecting structural performance should open an opportunity for the industries to further understand the potential benefit of RM technologies.

Acknowledgement
This project is sponsored by the Malaysia Automotive, Robotics and IoT Institute (MARii), previously known as Malaysian Automotive Institute. MARii is an agency under Ministry of International Trade
and Industry of Malaysia. This project is a part of Transportation Innovation Centre (TIC), a joint venture project between MARii and Swinburne University of Technology (SUT).

References
[1] ASTM International, *Standard Terminology for Additive Manufacturing – General Principles – Terminology*, in ISO / ASTM52900-15, ASTM International: West Conshohocken, PA, (2015)
[2] M. Aliakbari, *Additive Manufacturing State-of-the-Art, Capabilities, and Sample Applications with Cost Analysis*, in Department of Industrial Production, KTH Royal Institute of Technology: Sweden, (2012)
[3] K.P. Karunakaran et al., *Rapid manufacturing of metallic objects*. Rapid Prototyping Journal, 18(4): p. 264-280, (2012)
[4] T. Wohlers, *Wohlers Report 2012 : Additive Manufacturing and 3D Printing State of the Industry Annual Worldwide Progress Report*, Wohlers Associates, Inc, (2012)
[5] G. Dwivedi, S.K. Srivastava, and R.K. Srivastava, *Analysis of barriers to implement additive manufacturing technology in the Indian automotive sector*. International Journal of Physical Distribution & Logistics Management, 47(10): p. 972-991, (2017)
[6] G.N. Levy, R. Schindel, and J.P. Kruth, *Rapid Manufacturing and Rapid Tooling with Layer Manufacturing (Lm) Technologies, State of the Art and Future Perspectives*. CIRP Annals. 52(2): p. 589-609, (2003)
[7] J.G. Prada et al., *Direct digital manufacturing of an accelerator pedal for a Formula Student racing car*. Rapid Prototyping Journal, 22(2): p. 311-321, (2016)
[8] R. Limpert, *Brake Design and Safety*, SAE International, (2011)
[9] H. Gong et al., *Comparison of Stainless Steel 316L Parts Made by FDM- and SLM-Based Additive Manufacturing Processes*. JOM, (2018)