University of Baghdad
College of Engineering
JOURNAL OF ENGINEERING

Journal of Engineering

journal homepage: www.joe.uobaghdad.edu.iq

Volume 30 Number 1 January 2024

Improving Press Bending Production Quality through Finite Element Simulation: Integration CAD and CAE Approach

Erico Sofyan Chrissandhi[1,*], Eko Pujiyanto[2], Tonny A. Yuniarto[3]

[1,2]Department of Industrial Engineering, Faculty of Engineering, Universitas Sebelas Maret, Surakarta, Indonesia

[3]Engineering, Faculty of Industrial Technology, Universitas Atma Jaya Yogyakarta, Sleman, Indonesia

ericosofyanchrs@gmail.com[1], ekopujiyanto@ft.uns.ac.id[2], tonny.yuniarto@uajy.ac.id[3]

ABSTRACT

Efficient operations and output of outstanding quality distinguish superior manufacturing sectors. The manufacturing process production of bending sheet metal is a form of fabrication in the industry of manufacture in which the plate is bent using punches and dies to the angle of the work design. Product quality is influenced by plate material selection, which includes thickness, type, dimensions, and material. Because no prior research has concentrated on this methodology, this research aims to determine V-bending capacity limits utilizing the press bending method. The inquiry employed finite element analysis (FEA), along with Solidworks was the tool of choice to develop drawings of design and simulations. The ASTM E290 standard guides this study. The software in this package may combine CAD (Computer-Aided Design) and CAE (Computer-Aided Engineering) without requiring extra design applications. This study tested SPCC and SPHC plate materials with five thickness variations. The findings embrace the number of failure risks associated with press bending exhibited on the von Mises stress diagram, which is directly proportional to showing the thickness limit of each material type throughout the bending process. The study's findings lay the groundwork for improving manufacturing quality by lowering the number of faulty goods produced by trial and error. Because the maximum allowable die width is 12 mm, the thickness limit of the press bending process is 2 mm. However, due to the greater intensity of the SPCC material, it has a reduced defect rate compared to SPHC material.

Keywords: Press Bending, Computer-Aided Design, Computer-Aided Engineering, Finite Element Analysis, ASTM E290.

*Corresponding author

Peer review under the responsibility of University of Baghdad.

https://doi.org/10.31026/j.eng.2024.01.02

Article received: 08 /08/2023

Article accepted: 25/12/2023

Article published: 01/01/2024

تحسين جودة إنتاج ثني الصحافة من خلال محاكاة العناصر المحدودة :تكامل التصميم بمساعدة الكمبيوتر والنهج الهندسي بمساعدة الكمبيوتر

إريكو سفيان كريساندي[1]،*، إيكو بوجيانتو[2]، توني يونيارتو[3]

[1،2] قسم الهندسة الصناعية بكلية الهندسة، جامعة سيبيلاس ماريت، سوراكارتا ، إندونيسيا

[3] قسم الهندسة الصناعية بكلية التكنولوجيا الصناعية، يونيفيرسيتاس أتما جايا يوجياكارتا، سليمان ، إندونيسيا

الخلاصة

تتميز قطاعات التصنيع المتفوقة بالعمليات الفعالة والمخرجات ذات الجودة الفائقة .إن عملية تصنيع إنتاج الصفائح المعدنية هي شكل من أشكال التصنيع في صناعة التصنيع حيث يتم ثني اللوحة باستخدام اللكمات والقوالب حسب زاوية تصميم العمل . تتأثر جودة المنتج باختيار مادة الألواح ، والتي تشمل السُمك والنوع والأبعاد والمواد .نظرًا لعدم تركيز أي بحث سابق على هذه المنهجية ، فإن الغرض من هذا البحث هو تحديد حدود سعة الانحناء V باستخدام طريقة الانحناء بالضغط .استخدم التحقيق تحليل العناصر المحدودة (FEA) ، جنبًا إلى جنب مع Solidworks كانت الأداة المفضلة لتطوير رسومات التصميم والمحاكاة . يوجه معيار ASTM E290 هذه الدراسة .قد يجمع البرنامج الموجود في هذه الحزمة بين(CAD التصميم بمساعدة الكمبيوتر) و(CAE الهندسة بمساعدة الكمبيوتر)دون الحاجة إلى تطبيقات تصميم إضافية .اختبرت هذه الدراسة مواد ألواح SPCC و SPHCبخمسة اختلافات في السماكة .تتضمن النتائج عدد مخاطر الفشل المرتبطة بانحناء الضغط المعروض في مخطط الإجهاد الخاص بفون ميزس ، والذي يتناسب طرديًا مع إظهار حد سمك كل نوع من المواد خلال عملية الانحناء .تضع نتائج الدراسة الأساس لتحسين جودة التصنيع عن طريق تقليل عدد السلع المعيبة الناتجة عن التجربة والخطأ .نظرًا لأن الحد الأقصى المسموح به لعرض القالب هو 12 مم ، فإن حد السماكة لعملية ثني الكبس هو 2 مم .ومع ذلك ، نظرًا لزيادة كثافة مادة SPHC، فقد انخفض معدل العيوب فيها مقارنةً بمواد.SPCC

الكلمات المفتاحية: الانحناء الصحفي ، التصميم بمساعدة الكمبيوتر ، الهندسة بمساعدة الكمبيوتر ، تحليل العناصر المحدودة، ASTM E290

1. INTRODUCTION

Indonesia's manufacturing industry accounts for 4.6% of the global industrial sector **(Budiyanti, 2016)**. Efficiency and high production quality are required to become a top manufacturing industry **(Omran, 2013; Hoe and Mansori, 2018)**. As a result, Indonesia's manufacturing sector has been steady since 1990 **(Budiyanti, 2016).** According to statistics from Badan Pusat Statistik Indonesia, the manufacturing industry, particularly the machinery sector, employs 0.16% of the total 14.17% of Indonesians in various industrial disciplines **(Statistik, 2023)**. Based on this statistic, over 44 million people in Indonesia work in manufacturing, particularly machines.
ATMI Duta Engineering Surakarta is a growing industrial enterprise specializing in equipment and sheet metal fabrication. Trudisk 3030, Trulaser 3030, Trubend V130, and Trumark equipment assist the production process at ATMI Duta Engineering. In addition to

the automatic machinery, there are welding, finishing, and manual procedures machines. The firm manufactures souvenirs, replacement parts, and customized goods **(ADE, 2018)**. Cutting, shaping, coating, heat treatment, and casting are some of the manufacturing methods used by the organization. Sheet metal bending is a sheet metal production method used in the firm. The plate is bent to produce an angle according to the work design requirements using a punch as a press tool and a die as a holding and shaping tool **(Boljanovic, 2014; Jain and Sharma, 2016)**.

Sheet metal bending is classified depending on the bending shape of the plate, which includes v-bending, r-bending, u-bending, and hemming **(Abe et al., 2020; Hai et al., 2020)**. In a study by **(Hai et al., 2020)**, the v-bending shape is the most often employed plate formation in bending. The plate is bent away from the straight axis at an angle of 1° until 179° in v-bending **(Chrissandhi, 2020).** Press Bending is a common bending technique for v-bending sheet metal because it is easy and ideal for bending sheet metal**.**

The plate gets pushed by the punch between the supporting dies during the press bending procedure, which involves bending it at three different spots **(Boljanovic, 2014)**. Sheet thickness, sheet dimensions, kind of sheet material, sheet shape, compression force, and supporting equipment all influence the final product quality for the press bending process employing v-bending shapes on sheet metal **(Armunanto et al., 2018)**. The compatibility of the parameters will impact the manufacturing quality because the product will be damaged or defective **(Boljanovic, 2014; Armunanto et al., 2018)**. Tearing, spring back, and cracking are just a few examples of product defects caused by sheet metal bending procedures **(Ruchiyat et al., 2019; Sahu and Pradhan, 2020; Sukarman et al., 2020)**. A fracture or cracks in the bending results in damage to the workpiece plate **(Chrissandhi, 2020)**. Product defects owing to poor manufacturing quality must be discarded, which increases the cost of the manufacturing process.

ATMI Duta Engineering uses limit tests for material bending capabilities to minimize the effects of crack faults during manufacturing. The test is performed directly on the company's bending machine, utilizing the trial and error method to determine the parameters for manufacturing. This process generates faults in numerous early products, including cracks. Furthermore, the trial and error process has disadvantages, such as the requirement to raise the cost of test materials, machine utilization expenses, and uneven product quality. Simulation testing may be used to anticipate the limitations of the bending process that will result in fractures. Defects caused by cracks in the workpiece plate can be eliminated by modeling finite element analysis (FEA) for the bending process before being processed on the machine to enhance production quality on the factory floor **(Karuppuswamy et al., 2022)**. Previous researchers have successfully carried out several prior studies connected to using CAE software using the notion of bending process simulation with CAE software **(Thuillier et al., 2010; Armunanto et al., 2018; Saravanan et al., 2018; Miranda et al., 2018; Cheng et al., 2019; Lu et al., 2020; Kumar et al., 2021; Karuppuswamy et al., 2022)**. **(Thuillier et al., 2010; Armunanto et al., 2018)** completed a research on FEA material tests concentrating related to the bending industry. However, this study was undertaken for the hemming procedure and did not include press bending testing for the v-bending model. Although the outcome of this study is a search for spring back defects, numerous studies have also done material testing using FEA and focused on analyzing press bending for a v-bending model **(Saravanan et al., 2018; Karuppuswamy et al., 2022)**. Kumar et al. conducted material testing using Finite Element Analysis (FEA) to examine press bending. The testing primarily targeted u-bending contours and aimed to analyze how differences in bending angles impacted the formation of cracks in the test material **(Kumar**

et al., 2021). Other research related to FEA for bending press analysis has also been carried out, but the research focuses on solving the spring back problem by getting the correct numerical input **(Miranda et al., 2018; Lu et al., 2020).** Research using FEA has also been carried out focusing on spring back problems, and the test objects used are pipes and not plates **(Cheng et al., 2019)**.

This study focused on an innovation gap revealed in earlier research by applying optimization of design methods that relied on finite element analysis (FE) alongside Solidworks software under ASTM E290 standards to examine fractures in sheet metal to enhance the firm's production quality. Research conducted at ATMI Duta Engineering was carried out to overcome the limitation of press bending capabilities with limited equipment available in the company. Solidworks was chosen because it can be used as software for developing tools, testing materials, and doing FEA simulations, all inside the same package. Because it is simple to use in design and is widely utilized by educational institutions and businesses, Solidworks design software may be used to address these weaknesses. This software's design of 2-dimensional and 3-dimensional graphics may be immediately replicated with existing features. The ASTM E290 standard reference for bending tests for fracture analysis in sheet metal is used to design instruments, test materials, and simulation procedures. The simulation was performed at ATMI Duta Engineering and used changes in the material's kind and thickness based on the material's availability and the frequency of usage in the firm. The simulation's tools (punch and die) depend on the firm's availability. The FEA investigation determined the thickness limit of the material accessible at the firm. This material thickness restriction is intended to eliminate product failures during production, increase the quality of ATMI Duta Engineering production floors, and utilize the research as a global standard.

2. METHODOLOGY

2.1 Materials and Tools for Bending

This bending modeling procedure uses Steel Plate Cold Rolled Coiled (SPCC) and Steel Plate Hot Rolled Coiled (SPHC). The materials utilized as test materials in this study were chosen based on the frequency with which ATMI Duta Engineering Surakarta uses this material. The thickness of the material used in the bending simulation process is tailored to the firm's availability. The cold rolling process produces SPCC or white plate to reach the desired thickness and width. SPCC is frequently used to make food cans, furniture, and electrical appliances. SPCC has the advantages of being formable, weldable and having a low surface roughness **(Bui and Le, 2016; Krakatau Steel, 2023a)**. SPHC is a specific type of steel plate made by hot rolling. SPHC is known as a black plate because of the existing oil coating (HRC-PO or Hot Rolled Coil - Picked Oil). SPHC is frequently utilized in the automobile sector to build frame parts, brake components, wheels, clutch plates, tube frames, and compressors due to the material's excellent corrosion resistance **(Naipinij et al., 2022; Krakatau Steel, 2023b)**.

According to a corporate study, the materials utilized came from PT. Krakatau Steel, a material provider. **Table 1** presents data on the type, thickness, and material properties used in the bending simulation research, which was obtained from the vendor's specifications of the plate material used by the firm. SPCC material standards are based on the JIS G3141 standard **(Japanese Standards Association, 2011)**, which is SPHC material specifications are based on JIS G3131 **(Japanese Standards Association, 2010)**. The information in **Table 1** will be utilized to create a design model in Computer-Aided Design (CAD) software.

Table 1. Material Data for Bending Simulation Testing **(Krakatau Steel, 2023a, 2023b)**.

Plate Type	Plate thickness(mm)	Tensile Strength (N/mm^2)	Elasticity Modulus (N/mm^2)
SPCC	0.5	420	205000
	1.0		
	1.5		
	2.0		
	2.5		
SPHC	0.5	325	200000
	1.0		
	1.5		
	2.0		
	2.5		

Punch and die are the instruments used in the press bending production process. Dimensions, component material kinds, and the bending ability of the punch employed on the TruBend machine are all required data tools. The TruBend machine is a bending machine used by PT. ATMI Duta Engineering in the manufacturing department. The collected data tools will be utilized as a reference in developing models in CAD software. At the company, there are two methods of bending work, specifically air bending and press bending, that are methods frequently utilized in the fabrication of sheet metal industry **(Chrissandhi, 2020)**, and each of them has limited tools used considering the company is without all of the TRUMPF bending machine's tools. The instruments available at the firm for the press bending process are punches and dies with a width of 12 mm since the equipment available only reaches that width. The information data on the specifications of the tools utilized at PT. ATMI Duta Engineering is shown in **Figs. 1 and 2** from the TRUMPF engine developer's catalogue **(TRUMPF, 2018)**.

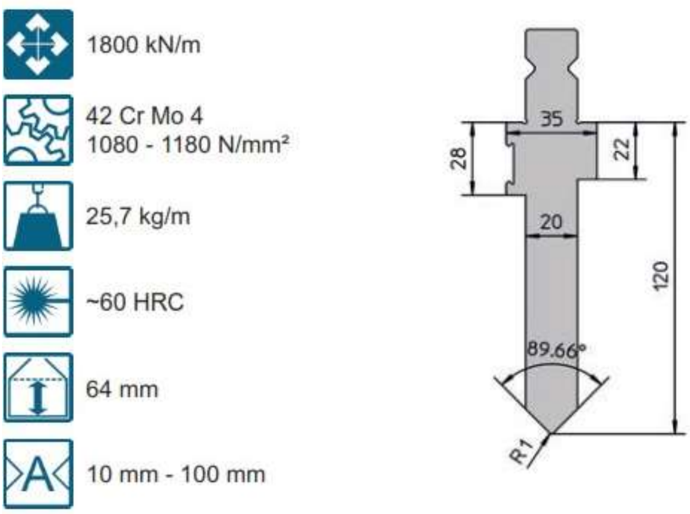

Figure 1. Punch used in the manufacturing process **(TRUMPF, 2018)**

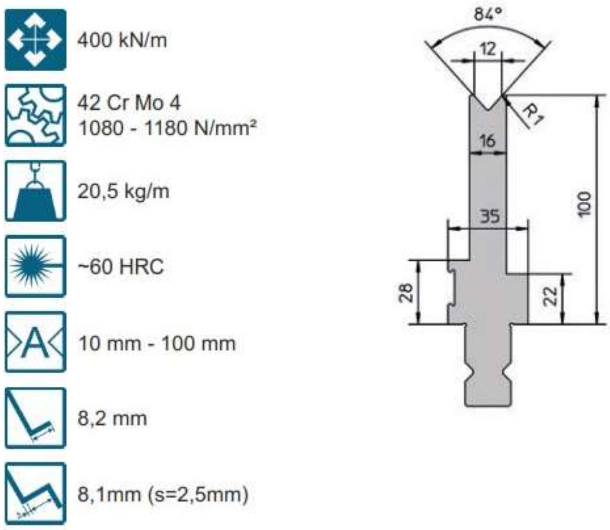

Figure 2. Dies are used in the manufacturing process **(TRUMPF, 2018)**

2.2 Bending Techniques

The most frequent and commonly used production method in sheet metal processing is sheet metal bending **(Saravanan et al., 2018)**. The side strength and moment of inertia increase throughout the bending process **(Jain and Sharma, 2016)**. This method is commonly used in the automotive and aviation sectors **(Boljanovic, 2014)**. Three-point bending is a type of plate bending involving 3 force locations during the bending process. The punch applies force to the plate as support at a position directly between the gaps of the die **(Al-Gasham, 2016; Bitzel et al., 2016; Armunanto et al., 2018)**.
The three-point bending method applies to press bending because the sheet metal has been bent with a punch tool along a straight path up to the bending result achieved according to the die used without support. Two sides of the die serve as supports, while one side of the punching tool is a suppressor. The highest sheet metal thickness that may be handled by press bending is 25 mm **(Bitzel et al., 2016)**. By multiplying the material's elasticity (E), inertia (I), and bottom dead point (Wo), Eq. (1) yields the bending force (F) for press bending or three-point bending **(Armunanto et al., 2018)**. The multiplication result has been divided by the measurement of the die width (L). The resulting bending force (F) will be used as the pressure input in the FEA simulation on materials with different material types and thicknesses.

$$F = \frac{48E*I*Wo}{L^3} \tag{1}$$

The bottom dead point (Wo) is the gap between the die's surface and the bottom of the die basin. The angle between the slanted side and the straight line has been determined from the die width and angle measurements; thus, this number may be computed using the trigonometry formula by applying tan. The moment of inertia (I), as seen in Eq. (2), is calculated by multiplying the material's thickness (t) by its width (w) and then dividing the result by 12. The bent rod's moment of inertia is divided by the nominal 12.

$$I = \frac{t^3*w}{12} \tag{2}$$

2.3 Design of Materials and Tools

The original data in Table 1, Figures 1 and 2, and the size, shape, and kind of material are used to create materials and tools into a 3-dimensional data model. This is done to obtain a decent simulation model near the findings under real-world situations. Solidworks software is used in the image design process to turn 2-dimensional photos into 3-dimensional model images. According to AppliCAD Indonesia **(AppliCAD Indonesia, 2014)**, the Solidworks software is CAD software manufactured by Dassault Systemes that industrial organizations and educational organizations extensively use due to the benefits of turning 2-dimensional designs into 3-dimensional ones. Solidworks features are easier to use than CAD applications for developing simple to complicated designs. Solidworks had three major menus when it first appeared: (1) Part is a menu for representing 3-dimensional things from 2 dimensions, and many characteristics in the part menu can be employed during the design process. (2) Assembly is a menu that combines various 3-dimensional photos from the components menu to create a product assembly. This feature includes finite element simulation to calculate compressive force and stress area since the simulation procedure requires more than one portion. (3) Drawing is a menu that converts three-dimensional pictures from the components and assembly menus into two-dimensional working drawings **(Chrissandhi, 2020)**.

2.4 Finite Element Analysis Simulation

The finite Element Method (FEM), commonly known as Finite Element analytical (FEA), is a technique for tackling different engineering issues using approximate solutions and numerical analytical tools **(Okereke and Keates, 2018; Liu et al., 2022)**. FEA may handle engineering issues such as strains, continuum mechanics, stresses, internal forces, and displacements of linear or non-linear structures. Because of the simplicity of its formulation, FEA is frequently employed **(Jagota et al., 2013; Liu et al., 2022)**. The advancement of computerized technology has facilitated FEA, with beneficial computer software for conducting this approach. FEM software includes Abaqus, ANSYS, MSC, Catia, Solidworks, and others. The software's design tools for developing goods or workpieces to be studied make FEA implementation easier **(Jagota et al., 2013; Okereke and Keates, 2018; Liu et al., 2022)**.

According to Arisma Data Setia, designs created in the Solidworks software program may be modeled without prior model construction **(Arisma Data Setia, 2018)**. The simulation tools in Solidworks software make study connected to The FEA of an end product or processes simpler because testing does not need resources for developing and researching a product. Static, design studies, fatigue, thermal (hot), buckling, frequency, drop testing, sub-modeling, pressure, non-linear, and linear dynamic simulations are all possible in Solidworks.

Non-linear simulations were employed for FEA on the press bending in this study. Non-linear simulations are utilized in FEA simulations that concentrate on the material used, as are non-linear simulations in other investigations **(Syed et al., 2016; Červinek et al., 2022; Zhang et al., 2022)**. Non-linear simulations are also utilized to determine the fracturing limitation of the material when rubbed, as are static loads, since bending loads do not vary or remain constant. The FEA simulation technique also relates to the ASTM (American Society for Testing and Materials) standard, which is a global testing and materials standard developed by ASTM International. The ASTM E290 standard evaluates finite element flexural characteristics employed in the 3-point bending procedure **(ASTM International, 2009)**. The testing specimen is in the shape of a rectangular plate. The specimen plate is 38 mm x 76 mm long and in breadth, with different thicknesses. The test specimen's bending angle was 90° **(ASTM International, 2009)**.

3. RESULTS AND DISCUSSION

3.1 Bending Force

To calculate the bending force (F), as indicated in Eq. (1), parameters that include the modulus of elasticity (E) for every material type are required, as shown in **Table 1**. In addition, the die's width data may be observed in **Fig. 1.** To determine the bending force (F), parameters that include bottom dead point (Wo) and inertia (I) must be computed. **Table 2** shows the results of the bending force (F) calculation for each material with changes in type and thickness. According to the findings of these calculations, the larger the bending force, the more dense the material. Based on the material, the SPCC material requires a stronger bending force (F) than the SPHC material, considering the SPCC material's elastic modulus (E) is greater than the SPHC, as shown in **Table 1.**

Table 2. Bending Force (F) or Tonnage (N) Calculation Results

Type of Plate	Plate Thickness (mm)	Tonnage (N)
SPCC	0.5	29.33
	1.0	234.60
	1.5	791.78
	2.0	1876.80
	2.5	3665.63
SPHC	0.5	21.99
	1.0	175.95
	1.5	593.83
	2.0	1407.60
	2.5	2749.22

3.2 Drafting Material Design and Tools

Design planning is carried out for study data tools and materials to be simulated within CAE software applications. The Solidworks program is used to create research tools and materials. The tools employed in punches and dies are intended to match the size of the components in the machine shown in **Figs. 1 and 2**. The shapes and sizes from **Figs. 1 and 2** are used to create a three-dimensional design in Solidworks software. As a research reference, the test material in the shape of a plate is given with parameters determined by ASTM E290 **(ASTM International, 2009). Fig. 3** shows the outcomes of the materials and tools' three-dimensional design.

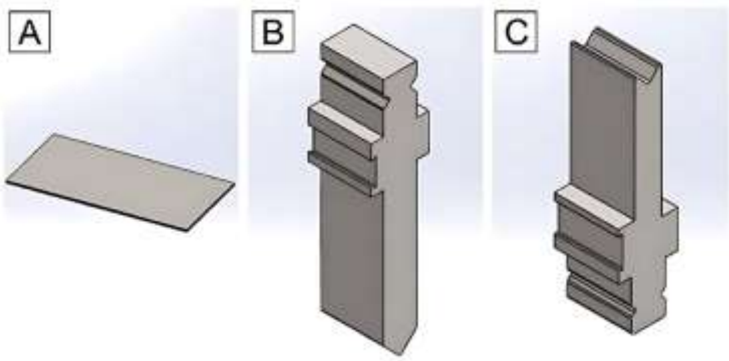

Figure 3. Outcomes of Design: (a) Materials (b) Punches (c) Die.

The structure of the components made of materials and tools is also changed to the original information in **Table 1, Figs. 1 and 2** by entering it into Solidworks. This is done to develop a credible simulation model that is near to the outcomes in real-world situations and can significantly enhance product quality. However, there are issues with entering material information for steel or plate materials throughout the design phase. The materials required for design are SPCC and SPHC, which are not accessible in the Solidworks program. The disparity in name standards with the variety of materials used by firms following ASTM and JIS naming standards causes this difficulty. Meanwhile, the word Solidworks is an abbreviation for the AISI standard. The answer to these variations is to determine the material based on the tensile strength and elastic modulus characteristics of the SPCC and SPHC materials stated in **Table 1**.

3.3 Finite Element Analysis Simulation

The simulation was performed using the same CAE program for developing tools and materials, Solidworks. This procedure uses Solidworks' simulation tool, with non-linear simulations picked from the sub-menu. In the simulation, contact set parameters for each side that will rub. The contact set was separated into two pieces for this investigation. The first section is the plate's top side, which will rub against the punch side, and the second is the plate's bottom side, which will rub against the die side. Fixed geometry parameters were used to establish which sections would not move throughout the simulation, and the punch's depth setting was reduced. In this test, the mesh density is at a fine level. The extremely fine mesh density has the maximum density enabled by the Solidworks program, which increases the accuracy of the simulation results.

Results from the finite element simulation will take the form of stress zones that develop during the bending process. The analysis of outcomes of simulations refers to the size of the stress region that happens through the simulation process as a whole. The green hue shows a normal stress region; the more red the color of the stressed area, the higher the chance of breakdown in the process of bending in the form of a fracture. Materials with stress sections with a broad red color spectrum at the bending angle are the sections that have the highest stress points during the bending procedure. The results of the red color spectrum in the material are still safe if no errors occur during the simulation. This error indicates that the material cannot be processed by press bending with the existing punch and dies so that cracks will continue to occur. The color dispersion will be aggregated into a unit amount that takes the shape of von Mises stress, representing the material's elastic limit through bending.

Bending FEA simulation results for each kind of material with thickness modifications are presented in **Figs. 4 and 5**.
Bending test findings revealed that the most important force in the press bending process occupies that region of the middle radius of the plate, which is pushed by the punch and die in the von Mises stress graph shown in **Figs. 4 and 5**. The existence of the highest value out of **Figs. 4 and 5** indicates the region of the radius's center. The amount of load dispersion is shown by the color changes that occur. The error in the computer simulation outcomes with a material thickness of 2.5 mm reveals that if the pressure applied is too great, the remedy is to widen the die. Changing the usage of wider dies will affect the force needed owing to die width within the three-point formula utilizing bending as a divider **(Armunanto et al., 2018)**. The greater the width of a die, the less force is required. **Table 3** is a comparison table based on the von Mises stress graphs in **Figs. 4 and 5** to facilitate the findings of the FE bending simulation testing for each kind of material with thickness changes.

Table 3. Results of Von Mises Stress from SPCC and SPHC Materials.

Thickness (mm)	Simulation Results	
	SPCC (N/mm^2)	SPHC (N/mm^2)
0.5	45815.6367	44694.4844
1.0	68495.2266	66699.1797
1.5	81202.6641	79733.4063
2.0	95891.4219	89871.5234
2.5	61966.9727	52774.7734

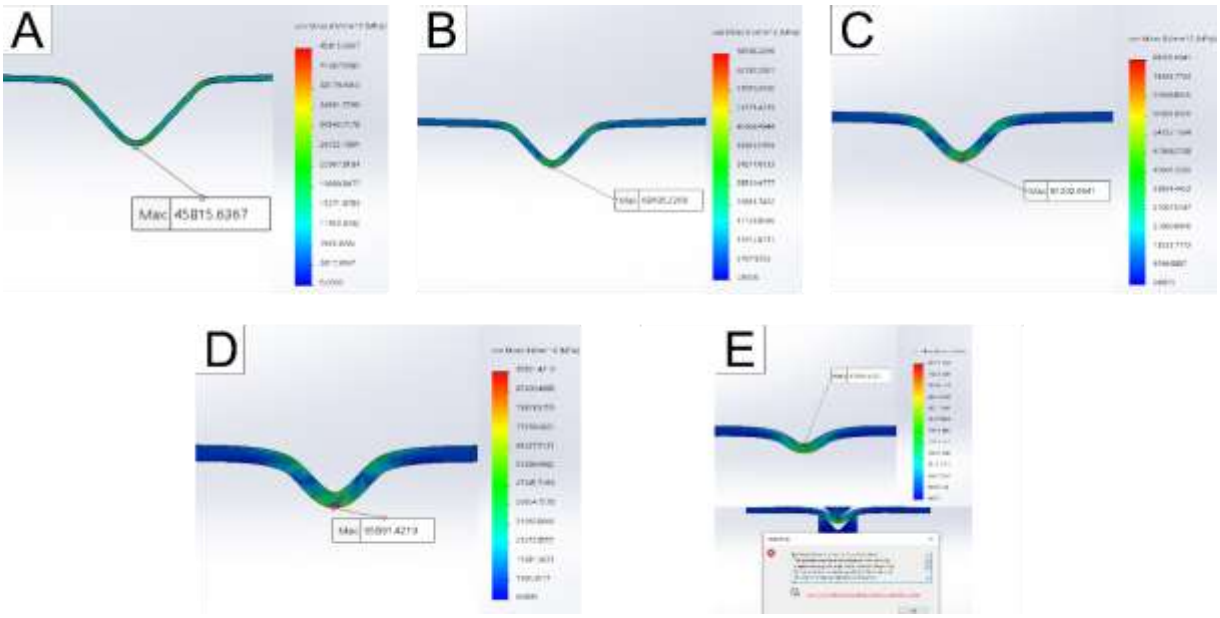

Figure 4. SPCC Finite Element Material Simulation Results for (a) Thickness 0.5 mm; (b) Thickness 1.0 mm; (c) Thickness 1.5 mm; (d) Thickness 2.0 mm; (e) Thickness 2.5 mm

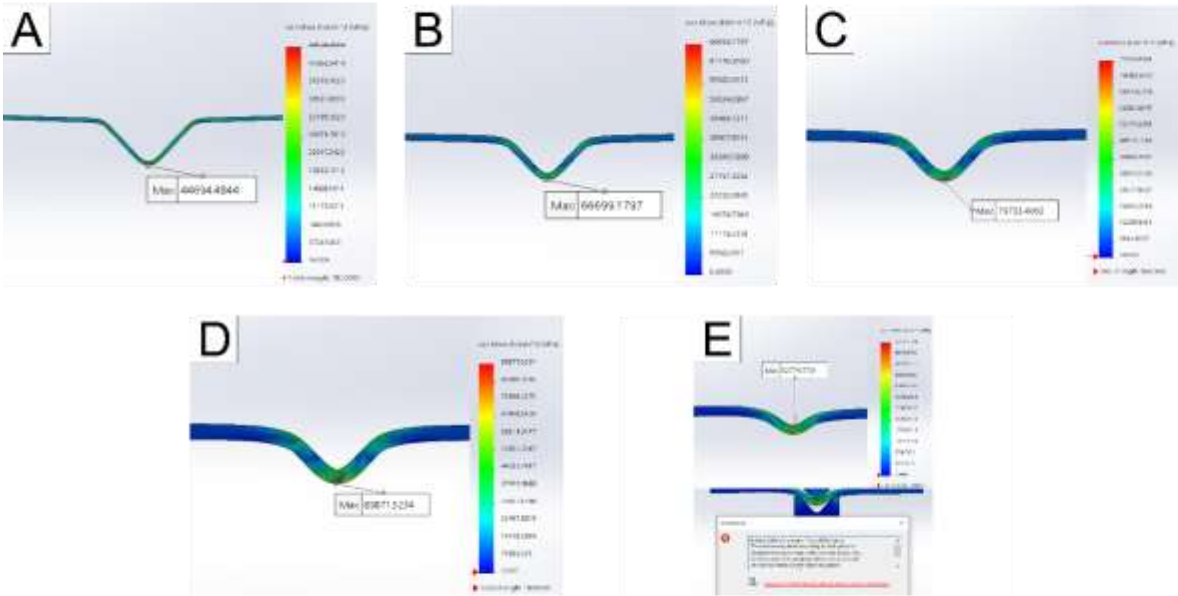

Figure 5. SPHC Finite Element material simulation results for the thickness of:
a) 0.5 mm; b) 1.0 mm; c) 1.5 mm; d) 2.0 mm; e) 2.5 mm

The von Mises stress value results from forecasting failure in material loaded with multiaxial stress. Failure in loading might manifest as cracks, fractures, or irreversible damage **(Putra et al., 2019)**, as demonstrated by the outcome of loading plates having a material thickness of 2.5 mm for each material. Based on the results of von Mises Stress from SPCC and SPHC in **Table 3**, the findings achieved for each plate material show that SPCC plates have more excellent resistance than SPHC plates. That result is indicated by the von Mises value produced by the SPCC material for each thickness being more significant than that produced by the SPHC material. The higher resistance of SPCC material compared to SPHC material in von Mises Stress results is due to the tensile strength and elastic modulus specifications of SPCC material, which are also higher than SPHC material, referring to **Table 1**. SPCC material has high resistance because SPCC material is used for the production of more durable goods (automotive, gas cylinders, etc.) compared to SPHC material (home furniture, electrical furniture, cans). Besides being constructed as a comparison table, the FE bending simulation results are compared with the curve lines in **Fig. 6**.

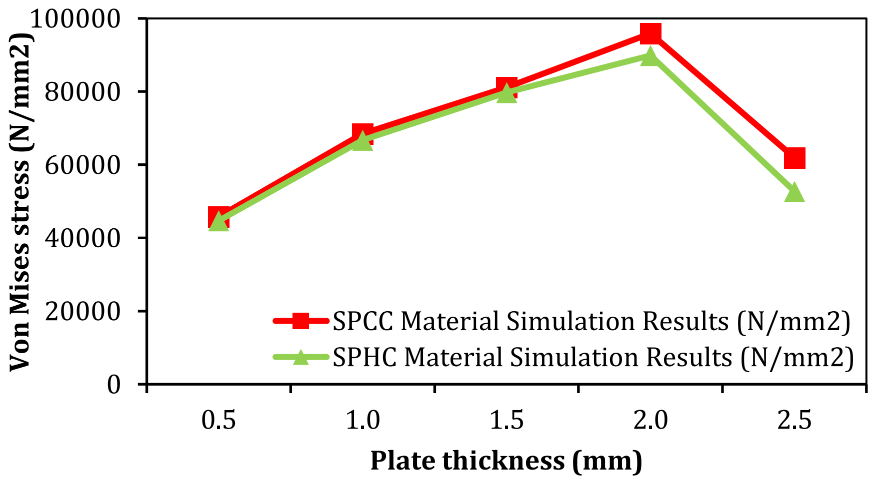

Figure 6. Graph of the Von Mises Stress Comparison Curve for SPCC and SPHC.

The form of the comparison curve in **Fig. 6** illustrates that the potential of failure for each material increases in precisely proportion to the thickness of the plate to be treated by press bending. Although at a material thickness of 2.5 mm, the lines for both materials have shrunk, and the simulation outputs are inaccurate. As illustrated in the graph for the fatigue limit, the reduction in the contour of the curve and the error suggest a fracture has developed. In numerous additional investigations, a descending slope indicates the presence of material flaws **(Armunanto et al., 2018; Chrissandhi, 2020)**. As a result, for the company's high press bending production quality, a material thickness of 2.5 mm is the thickness limitation that may be produced for bending for dies with a width of 12 mm.

4. CONCLUSIONS

The press bending procedure is successful if there are no failures in the bending press outcomes, such as cracks. Several parameters, such as the punch radius and die width, can cause cracks in the material due to the sheet metal bending process. The small punch and die equipment employed in the study site, which was 12 mm wide, proved to be an impediment. Based on the study of the press bending test performed in Solidworks using non-linear simulation, the capacity limit applicable to SPCC and SPHC materials is 2.5 mm according to the maximum capacity of press bending tools available at ATMI Duta Engineering Surakarta manufactures punches and dies with a width of 12 mm.
The materials utilized in the test, SPCC and SPHC, are the most often used bending materials. According to the study, the SPHC material is less likely to fail when bent than the SPCC. However, regarding the danger of process failure during press bending, the SPHC material outperforms the SPCC material. This is demonstrated by the von Mises value of the SPHC material for each thickness less than or equal to that of the SPCC material because the SPHC material must have excellent resistance to have qualities that are more difficult to form than the SPCC material. In the comparison calculation, if the risk value acquired by the SPHC is regarded to be zero, the average percentage of the five thicknesses against failure of the press bending process compared to the SPCC material is 6.23%. This percentage indicates that the press bending process failure in the form of a fracture on the 6.23% SPCC material has greater strength than the SPHC material. Using finite element simulation research in the current study can replace the trial and error procedure at ATMI Duta Engineering Surakarta. The replacement of this approach aids in the improvement of the firm's manufacturing quality, allowing it to become an excellent company. The simulation approach used will decrease product flaws caused by trial and error. Reduced crack faulty items can reduce production expenses in the firm, allowing production costs to be reduced as well. The results of this research only refer to the availability of materials often used in the company and the availability of bending equipment used in the bending press production process in the company. Further research to find the thickness limits of the press bending process with varying materials and bending equipment can be applied to obtain results from a different point of view.

REFERENCES

Abe, Y., Ijichi, W., Mori, K., and Nakagawa, K., 2020. Hemming with Pre-Bent Inner Sheet for Joining Ultra-High Strength Steel Sheets of Automobile Parts. *Journal of Manufacturing and Materials Processing*, 4(77), pp. 1–12. Doi:10.3390/jmmp4030077

ADE., 2018. ATMI Duta Engineering hi-tech sheet metal fabrication with laser technology : Products. https://ade.co.id/ (accessed Jan. 05, 2023)

Al-Gasham, T.S.S., 2016. Experimental Behavior of Steel-Concrete-Steel Sandwich Beams with Truss Configuration of Shear Connectors. *Journal of Engineering*, 22(4), pp. 18–29. Doi:10.31026/j.eng.2016.04.02

AppliCAD Indonesia., 2014. Solidworks Software Functions. AppliCAD Indonesia Co., Ltd. https://applicadindonesia.com/fungsi-software-solidworks/ (accessed Feb. 25, 2023)

Arisma Data Setia., 2018. Solidworks Simulation. https://arismadata.com/solidworks/products-simulation.php (accessed Feb. 25, 2023)

Armunanto, V.B., Chrissandhi, E.S., Yosef, H., Hernanto, K.G., Herdianto, P.S., and Kurniawan, Y.A.D., 2018. Hemming sheet metal analysis with variation in material type and thickness. *Journal Material Dan Proses Manufaktur*, 2(2), pp. 136–144. Doi:10.18196/jmpm.2230

ASTM International, 2009. Designation : E 290 - 14 - Standard test methods for bend testing of material for ductility. Doi:10.1520/E0290-14

Bitzel, H., Borcherdt, J., MÜller, J., Neidhart, F., Parey, K., Rau, A., Riecke, S., Schmid, A., Trentmann, G., Vorländer, G., and Zimmermann, K., 2016. *The fascinating world of sheet metal*, (Bending-sheet metal processing with corners and edges). pp. 100–111, TRUMPF GMBh.

Boljanovic, V., 2014. *Sheet metal forming processes and die design*. New York, USA: Industrial Press Inc.

Budiyanti, E., 2016. *Reinforcement of Indonesia's manufacturing industry*. Pusat Penelitian Badan Keahlian DPR RI, pp. 13–16. http://berkas.dpr.go.id/puslit/files/info_singkat/Info Singkat-VIII-12-II-P3DI-Juni-2016-50.pdf

Bui, A.H., and Le, H., 2016. Strength and microstructure of cold-rolled if steel. *Acta Metallurgica Slovaca*, 22(1), pp. 35–43. Doi:10.12776/ams.v22i1.690

Červinek, O., Pettermann, H., Todt, M., Koutný, D., and Vaverka, O., 2022. Non-linear dynamic finite element analysis of micro-strut lattice structures made by laser powder bed fusion. *Journal of Materials Research and Technology*, 18, pp. 3684–3699. Doi:10.1016/j.jmrt.2022.04.051

Chrissandhi, E.S., 2020. Aplikasi finite element untuk optimasi proses desain press bending di PT. Atmi Duta Engineering Surakarta Berbasis Software CAE. Thesis, Atma Jaya Yogyakarta University.

Cheng, X., Guo, X., Tao, J., Xu, Y., El-Atry, A. A., Liu, H., 2019. Investigation of the effect of relative thickness (t0/d0) on the formability of the AA6061 tubes during free bending process. *International Journal of Mechanical Sciences*, 160(2019), pp. 103-113. Doi:10.1016/j.ijmecsci.2019.06.006

Hai, V.G., Minh, N.T.H., and Nguyen, D.T., 2020. A study on experiment and simulation to predict the spring-back of SS400 steel sheet in large radius of V-bending process. *Material Research Express*, 7(1), pp. 1–15. Doi:10.1088/2053-1591/ab67f5

Hoe, L.C., and Mansori, S., 2018. The effects of product quality on customer satisfaction and loyalty: evidence from Malaysian engineering industry. *International Journal of Industrial Marketing*, 3(1), pp. 20–35. Doi:10.5296/ijim.v3i1.13959

Jagota, V., Sethi, A.P.S., and Kumar, K., 2013. Finite Element Method: An overview. *Walailak Journal of Science and Technology (WJST)*, 10(1), pp. 1–8. Doi:10.2004/wjst.v10i1.499

Jain, V., and Sharma, P., 2016. A study of strength analysis in sheet metal by bending process: A review. *International Journal of Engineering Sciences & Research Technology*, 5(9), pp. 55–60. Doi:10.5281/zenodo.61450

Japanese Standards Association., 2010. JIS G 3131 : Hot-rolled mild steel plates, sheet and strip (Patent No. JIS G 3131 : 2010 (E)). https://www.jfe-steel.co.jp/en/products/sheets/catalog/b1e-001.pdf

Japanese Standards Association., 2011. JIS G 3141 : Cold-reduced carbon steel sheet and strip (Patent No. JIS G 3141 : 2011 (E)). https://www.modiranahan.com/uploads/pdf/JIS-G-3141-2011.pdf

Karuppuswamy, P., Bhagyanathan, C., Sathish, S., and Elangovan, D., 2022. Comparisonal analysis of "V" punches with various radii and its impact on the steel material used for sheet metal operations. *Materials, Design and Manufacturing for Sustainable Environment*, pp. 113–121. Doi:10.1007/978-981-19-3053-9_10

Kumar, K., Bhadauria, S.S., Singh, A.P., 2021. Finite element analysis of edge and cleavage cracks in U-bend specimen using contour integral approach. *Materials Today: Proceedings*, 44(1), pp. 865-870. Doi:10.1016/j.matpr.2020.10.789

Krakatau Steel., 2023a. Cold Rolled Coil/Sheet. https://www.krakatausteel.com/viewcontent/18

Krakatau Steel., 2023b. Hot Rolled Coil/Plate. https://www.krakatausteel.com/viewcontent/18

Liu, W.K., Li, S., and Park, H.S., 2022. Eighty years of the finite element method: birth, evolution, and future. archives of computational methods in engineering, 29, pp. 4431–4453. Doi:10.1007/s11831-022-09740-9

Lu, K., Luo, J., Zou, T., Li, D., Peng, Y., 2020. Axial formation and cross-section distortion control for stretch bending based on batch simulations. I*OP Conference Series: Materials Science and Engineering*, 967, pp. 1-13. Doi:10.1088/1757-899X/967/1/012038

Miranda, S.S., Barbosa, M.R., Santos, A.D., Pacheco, J.B., Amaral, R.L., 2018. Forming and springback prediction in press brake air bending combining finite element analysis and neural networks. *Journal of Strain Analysis for Engineering Design*, 53(8), pp.1-18. Doi:10.1177/0309324718798222

Naipinij, S., Sukieum, S., Namprai, R., and Nilsonthi, T., 2022. To, formation of thermal oxide scale and its adhesion Strip, hot-rolled low carbon steels with different final thicknesses. E3S Web of Conferences 355. Doi:10.1051/e3sconf/202235502008

Okereke, M., and Keates, S., 2018. *Computational Mechanics and the Finite Element Method.* Springer International Publishing. Doi:10.1007/978-3-319-67125-3_1

Omran, S.H., 2013. Development of quality rating evaluation of outgoing product case study applied at the general company for vegetable oils. *Journal of Engineering*, 19(4), pp. 67–82. Doi:10.31026/j.eng.2013.04.10

Putra, Y.E.A., Anggoro, P.W., Yuniarto, T., Bayuseno, A.P., Jamari, and Bawono, B., 2019. Optimization of mechanical parameters on outsole shoes orthotic comfort using finite element analysis. *International Biomedical Instrumentation and Technology Conference* (IBITeC), pp. 31–36. Doi:10.1109/IBITeC46597.2019.9091719

Ruchiyat, A., Anhar, M., Yusuf, and Polonia, B.S.E., 2019. The effect of heating temperature on the hardness, microstructure and v-bending spring back results on commercial steel plate. *Journal of Applied Engineering and Technological Science*, 1(1), pp. 1–16. Doi:10.37385/jaets.v1i1.10

Sahu, Y.K., and Pradhan, M.K., 2020. Modelling and simulation of deep drawing process of circular cup on AL1200 using finite element analysis. *Advances in Simulation, Product Design and Development*, pp. 29–42. Doi:10.1007/978-981-32-9487-5_3

Saravanan, S., Saravanan, M., and Duraisamy, J., 2018. Study on effects of spring back on sheet metal bending using simulation methods. *International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)*, 8(2), pp. 923–932. Doi:10.24247/ijmperdapr2018106

Statistik, B.P., 2023. Proporsi Tenaga Kerja pada Sektor Industri Manufaktur (Persen) 2019-2022. https://www.bps.go.id/indicator/9/1217/1/proporsi-tenaga-kerja-pada-sektor-industri-manufaktur.html

Sukarman, S., Anwar, C., Rahdiana, N., Khoirudin, K., and Ramadhan, A.I., 2020. Analysis of the effect of radius dies on springback of stainless-steel sheet metal in the v-die hydraulic bending process. *Jurnal Teknologi*, 12(2), pp. 123–132. Doi:10.24853/jurtek.12.2.123-132

Syed, I.S., Mohamed, O.A., and Rahman, S.A., 2016. Non-linear Finite Element Analysis of Offshore Stainless Steel Blast Wall under High Impulsive Pressure Loads. *Procedia Engineering*, 145(2016), pp. 1275–1282. Doi:10.1016/j.proeng.2016.04.164

Thuillier, S., Maoût, N.Le, and Manach, P.Y., 2010. Bending limit prediction of an aluminium thin sheet. *International Journal of Material Forming*, 3(1), pp. 223–226. Doi:10.1007/s12289-010-0747-7

TRUMPF., 2018. Bending Tools : TruBend Cell 7000. TRUMPF Maschinen Austria GesmbH & Co. Kg.

https://www.trumpf.com/filestorage/TRUMPF_Master/Products/Services/01_brochures/TRUMPF-bending-tools-catalog-TruBend-Cell-7000-EN.pdf

Wang, J., Verma, S., Alexander, R., and Gau, J.T., 2008. Springback control of sheet metal air bending process. *Journal of Manufacturing Processes,* 10(1), pp. 21–27. Doi:10.1016/j.manpro.2007.09.001

Zhang, A., Xie, Z., Shi, L., Zhang, Y., Zhou, D., and Zhang, X., 2022. Research on the Simplified method of non-linear finite element analysis for CFS-SPR connections. Research on the Simplified Method of Non-linear Finite Element Analysis for CFS-SPR Connections, 12(1925), pp. 1–20. Doi:10.3390/buildings12111925