#694

Design of Progressive Die Sequence by Considering the Effect of Friction, Temperature and Contact Pressure

Esmeray Üstünyagiz[1, a *] and Taylan Altan[1, b]

[1]Center for Precision Forming, The Ohio State University, 339 Baker Systems, 1971 Neil Avenue, Columbus, OH 43210, USA

[a]ustunyagiz.1@osu.edu, [b]altan.1@osu.edu

Keywords: Multi-stage forming, process design, progressive die sequence, temperature, friction.

Abstract. Progressive and transfer dies are used for forming of sheet metal parts in large quantities. For a given part, the design of progressive die sequence involves the selection of the number of forming stages as well as the determination of the punch and die dimensions at each stage. This design activity is largely experience-based and requires prototyping involving several trial and error operations. In some cases, empirical data and the experience based design procedure can be combined with Finite Element Method (FEM) based analysis to reduce time and cost. Often, when using FEM in progressive die design, friction and its effect upon temperatures is not adequately considered. However, at each forming station the plastic deformation and the tribological conditions influence the material flow as well as the temperatures and pressures at the tool/workpiece interface. The performance of the lubricant and coolant, used in progressive die forming, is affected significantly by interface pressure and temperatures. Therefore, a progressive process and die design methodology should include the consideration of metal flow as well as temperatures and pressures. Heat transfer coefficient, friction, plastic deformation, forming speed at each forming stage, time for part transfer from one stage to the next, and the ability of the used lubricant to cool the dies, have considerable effect upon a successful stamping.

This paper describes a method for designing a progressive die sequence for forming axisymmetric sheet metal parts. The methodology for process sequence design combines experience based empirical data obtained through previous designs, design rules and numerical simulations including plastic deformation and friction. The initial experience-based design was refined using FEM and the thinning of the material in each successive drawing stage was calculated. The thermo-mechanical model was obtained using a constant friction coefficient along the tool/workpiece contact zone. Finally, the tool/workpiece interface temperature and the normal pressures were estimated in order that the lubricant can be selected based on these process conditions. The design predictions, made by using empirical data and FEM, were compared with experimental data.

Introduction

One of the most challenging issue in progressive die design is to determine the minimum number of the required stations to obtain the desired final geometry and surface finish. Such a study must also include design variations in process parameters.

A knowledge based system combining artificial intelligence technique and CAD system [1] has been proposed to automatically determine the process sequence. However, such systems can handle only the limited given knowledge based process conditions. For the prediction of number of operations and tooling geometry of multi-stage deep drawing, full analysis of each step is necessary. With the developments in computer based FE calculations, many researchers studied the full design sequence to predict the applicability of the design of the desired part [2], investigated limitations that occur during forming [3] and optimized the process sequence to reduce the forming steps [4].

However, these studies do not include a general guideline with sensitivity to process parameters such as temperature, and contact pressure, which are especially critical due to their direct effect on the lubricant performance.

The partner company of this project utilizes past experience and experimental die tryouts using prototyping to design the process sequence. The objective of this study is to develop a FE based strategy for designing a progressive die sequence for forming axisymmetric sheet metal parts (Part B, shown in Fig. 1b). The design also includes the estimation of tribological parameters such as tool/workpiece interface pressures and temperature.

Design Strategy

In order to develop an initial design strategy, an example part, Part A (Fig. 1a), was investigated using commercial implicit FEM code, DEFORM 2D. The parameters for the numerical model was taken from the previous study [5].

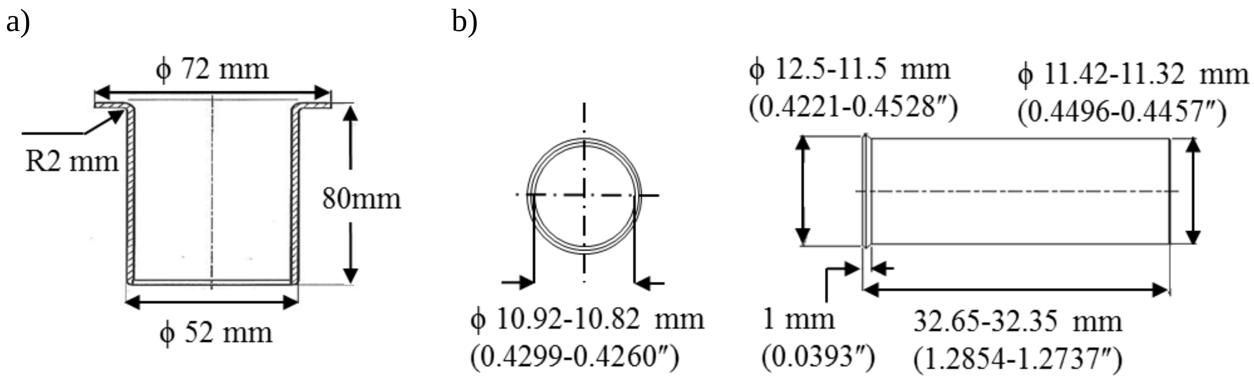


Figure 1. Geometry of a) example existing part (Part A, AISI 1008), and b) current part (Part B, AISI 305).

Maximum wall thinning after each drawing stage as well as the punch diameters are illustrated in Fig 2. In production values were obtained after simulating all the stages using the geometrical parameters given by the industrial partner. For the same process parameters used in previous study and current study, the maximum thinning differs from Stage 6. The reason may be related to the version of FE code. Furthermore, the previous study was conducted in inches. Small differences in the input resulted from the unit convergence and/or the code may cause slight differences in the thinning. The allocation of the variations may lead to a larger difference as forming continues. At the end of the Stage 10, the wall thinning remains below 30 %, which is assumed as the threshold value before the fracture for carbon steel, based on experience at Center for Precision Forming (CPF).

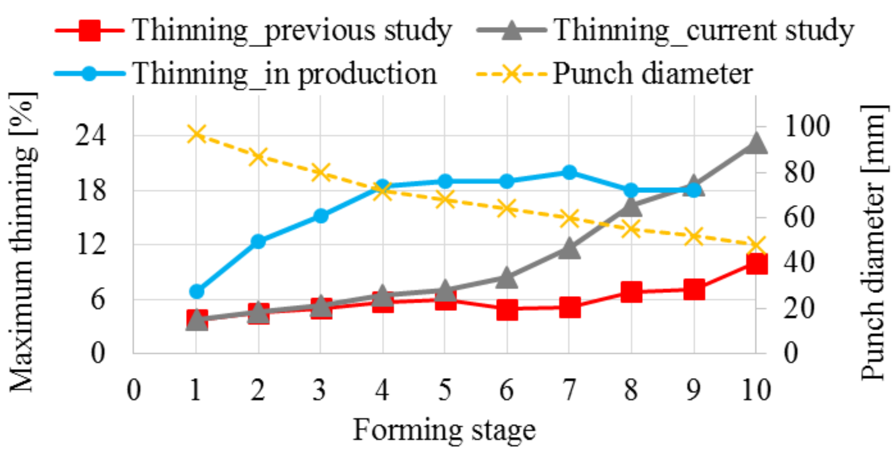


Figure 2. Maximum thinning and punch diameter with respect to forming stages for existing part, Part A.

In addition to the modellig knowledge, other design guidelines obtained from the investigation of existing part are as follows

- According to simulation results, for the first stage, wall thinning is restricted to less than 4 % of the sheet thickness. 4 % thinning constraint will be used for forming the new part.
- The results suggest that the punch diameter of the second drawing should not hit the part at the location where the maximum thinning after first drawing occurs. This information will be taken into account.

Process Sequence Design for the New Part (Part B) from AISI 305 Stainless Steel

Determination of First Drawing Stage Parameters. The final part geometry was given in Fig. 1b. First stage of the drawing is considered to be the most important one as the entire forming sequence depends upon it. The design involves determination of punch and die diameters, die/punch clearance, punch corner radius, die corner radius, drawing depth and blank holder force (BHF).

The inner diameter reduction in the first stage was suggested by the industrial partner to be 41.5 %. For the given reduction and initial blank diameter (49.53 mm), the punch diameter is 28.98 mm.

The clearance is usually in the range of t_0-1.2t_0 where t_0 is the initial sheet thickness [6]. In this study, clearance was selected as 1.15t_0. As a result, for initial sheet thickness t_0=0.254 mm, die dimeter of the first drawing stage was $d_d=d_p+2C$= 29.56 mm where d_p is the punch diameter and C is the die/punch clearance.

Recommended die corner radius is 8t_0-10t_0 [1]. As an initial assumption, die corner radius was chosen as 9t_0=2.29 mm.

According to the previous design outline, the ratio of punch corner radius to die corner radius was initially selected as 0.9. The punch corner radius for the first stage was therefore taken as 2.06.

The BHF is estimated from the following equation [7]

$$BHF = 10^{-3} c\left((DR-1)^3 0.005 D_0/t_0\right) S_u A_{BH} \quad (1)$$

where c is the empirical factor ranging from 2 to 3, DR is drawing ratio and equal to $DR=d_i/d_{(i+1)}$ where d_i is the part diameter for i'th step. S_u is Ultimate Tensile Strength (UTS) of the workpiece material, A_{BH} is the area where BHF applies. For AISI 305 stainless steel material the suggested BHF was found as 1.9 kN when c=2.

Using volume constancy, drawing depth was found as 13.83 mm when the cup is drawn until the outer edge of the blank enters the die corner radius.

Numerical Model. Numerical model used for the first drawing operation is shown in Fig. 3a. The geometrical parameters were suggested in previous sub-section. Punch, die and blankholder were modelled as rigid and blank as plastic material. The blank has 6 elements along the thickness direction and 3744 elements in total.

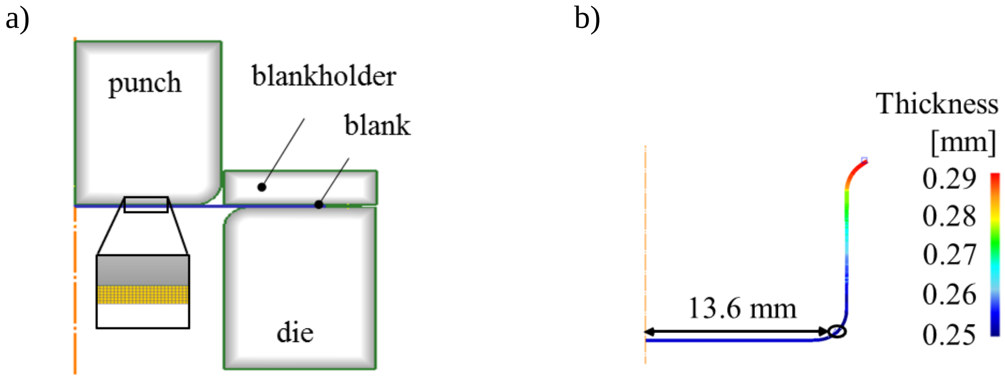


Figure 3. Illustration of the a) initial numerical model, and b) location of maximum thinning at the end of the first drawing stage calculated with optimized process parameters.

The blank material is AISI 305 stainless steel. Punch speed was assumed as 150 mm/s and coefficient of friction (cof) was 0.1. Stress-strain relation of blank material was $\bar{\sigma} = 1259\bar{\varepsilon}^{0.27}$ [MPa] based on bulge and dome test performed at CPF. For BHF 1.9 kN, maximum thinning exceeds 30%, the threshold value. The overestimation of BHF is because Eqn. 1 was most probably suggested for mild steel, whereas the material used for precision sleeve is stainless steel. The numerical analysis was therefore conducted with BHF 750 N.

Parameter Study. Parameter study was performed for the first drawing stage. The maximum thinning was constrained to 4 %. Table 1 shows the simulation matrix used to determine the optimum die corner radius. The values were selected within the recommended range [1]. For the production of precision sleeve, die corner radius is not critical, as the desired final shape does not have any flange with a defined radii (See Fig. 1b). Die corner radius $10t_0$=2.54 gives the least maximum thinning, 3.54 %, hence for the further analysis, die corner radius during the first drawing stage was 2.54 mm.

Table 1 Simulation matrix to determine optimum die corner radius.

Forming Stage #	Punch Diameter [mm]	Die Diameter [mm]	Die Corner Radius [mm]	Punch Corner Radius [mm]	Maximum Thinning [%]
1	28.98	29.56	*8t* = 2.03	1.83	4.33
	28.98	29.56	*9t* = 2.29	2.06	3.94
	28.98	29.56	10*t* = 2.54	2.29	3.54

Table 2 shows the simulation matrix used for the selection of punch corner radius to die corner radius ratio. When the ratio is 0.9 and 0.95, maximum thinning is 3.94, very close to the upper design limit 4 %. Increasing the ratio to 1.1 results in less thinning and was therefore selected.

Table 2 Simulation matrix to determine optimum punch corner radius/die corner radius ratio.

Forming Stage #	Punch Diameter [mm]	Die Diameter [mm]	Die Corner Radius [mm]	Punch Corner/ Die Corner Radius	Punch Corner Radius [mm]	Maximum Thinning [%]
1	28.98	29.56	2.54	0.9	2.29	3.94
	28.98	29.56	2.54	0.95	2.41	3.94
	28.98	29.56	2.54	1.1	2.79	3.15

The numerical analysis of the first stage was repeated with updated estimated process parameters. It was found that maximum thickening is about $1.16t_0$ within the range $1.1t_0$-$1.2\ t_0$ [6] meaning that there is probably no risk of wrinkling and ironing. Maximum thinning is 3.15 % below the threshold value of 4 %, located 13.6 mm from center as shown with an ellipse in Fig. 3b. The location of the maximum thinning in the first stage determines the maximum punch radius for the second stage. Because any value equal or bigger than 13.6 mm punch radius will result in hitting the maximum thinned zone.

Determination of Progressive Die Sequence. The process sequence of the precision sleeve was defined by the industrial partner as six drawings and one ironing station. Based on given part diameter reduction and initial blank diameter, the drawing depth of the each station was calculated from volume constancy. The assumptions made throughout the study are listed below.

- For the first two stages, the cup is drawn until the flange enters to the die corner. Following four stages (stage 3, 4, 5 and 6) take the flange formation into account.
- Die corner radius was reduced by a factor of 0.8 for subsequent stages [6].

- Similar to the previous part design, punch corner radius to die corner radius ratio was increased gradually by a factor of 1.2.

Results and Discussion

According to the listed assumptions, the geometric process parameters used for six drawing stages were calculated and corresponding simulations were conducted. The stress-strain distribution of sheet was saved as output after each forming stage and used as an input for the subsequent drawing.

Fig. 4a illustrates the deformed part shape after each forming stage. The ellipses represent the region where maximum thinning occurs. Maximum thinnings with respect to forming stages are presented in Fig. 4b. At the end of the forming stage 6, maximum thinning was around 20 % which is below the experienced based assumed threshold fracture value of 30 % for AISI 305 stainless steel.

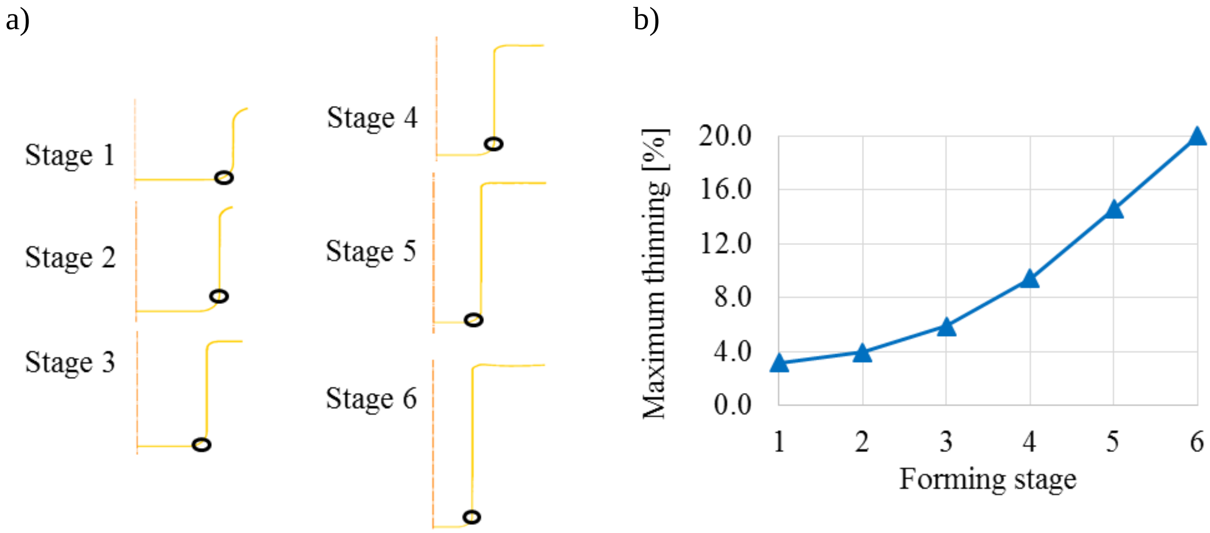


Figure 4. a) Deformed part geometry after each forming stage with the detail of maximum thinning region, illustrated with an ellipse and b) maximum wall thinning as a function of forming stage.

At the final stage of the forming sequence, ironing is applied to achieve uniform wall thickness and to reduce the inhomogeneity in thickness along the wall which may occur a result of drawing operations. For the final wall thickness 0.25 ± 0.05 mm and the maximum wall thickness 0.27 mm in the beginning of the ironing operation, the maximum wall reduction was calculated as 8 %, using the following equation: $r = (t_{i+1} - t_i)/t_i$ where t_i is the thickness at the i'th stage.

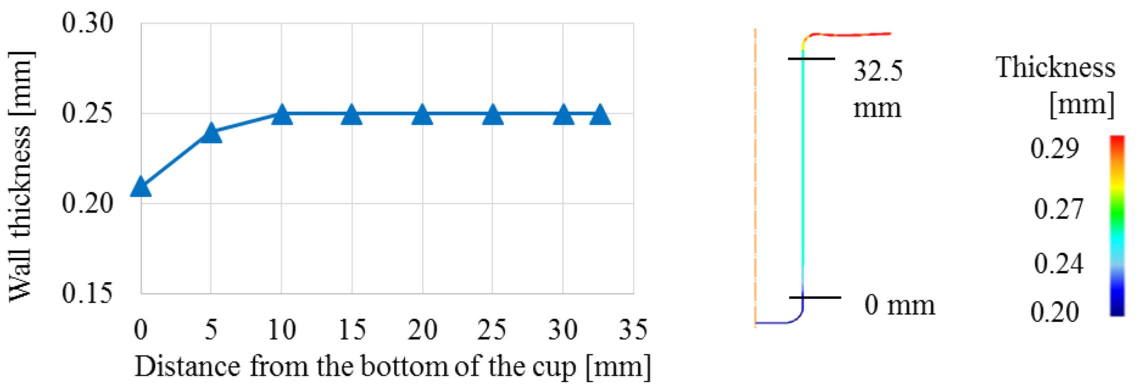


Figure 5. Wall thickness distribution of the ironed part with respect to distance from the bottom of the cup.

The wall thickness along the ironed tube with respect to distance from the bottom of the cup, given in Fig. 5, shows that the calculated thickness distribution is in the range of defined part tolerances 0.25 ± 0.05 mm.

Effect of Friction, Temperature and Pressure on Process Design

Numerical Model. Tool/workpiece interface temperature and normal pressures are critical tribological parameters to be considered for the lubricant selection. For determination of these parameters, thermal coupling of the process sequence was developed on top of the previously given mechanical model. The process parameters are summarized in Table 3. The temperature distribution of the sheet after each drawing operation was used as an input for subsequent forming operation.

Table 3. Process parameters used for the thermo-mechanical numerical analysis [8].

		Workpiece AISI 305	Die, Punch Vanadis 4
Young's modulus	[GPa]	200	225
Poisson's ratio		0.3	0.3
Structural density	[g/m^3]	7.9	7.6
Heat conductivity	[W/m·°C]	15	15
Heat capacity	[J/kg·°C]	500	460
Heat transfer coefficient (HTC) (assumed)	[kW/m^2·°C]	40	40
Initial temperature (assumed)	[°C]	20	20

Parameter Study. The first drawing stage was analysed futher to understand the effect of several parameters.

Firstly, contribution of the frictional heating was investigated. For that, maximum blank temperature were calculated for cof=0 and 0.1. Fig. 6 shows that when the frictional heating is neglected, i.e; cof=0, temperature along the strip cross section is relatively constant, around 84 °C. When cof is 0.1, the maximum temperature increases towards the die surface and reaches 101 °C due to heat generation that arises from friction. For the present study, cof was taken as 0.1, which gives the maximum lower die contact temperature as 39°C.

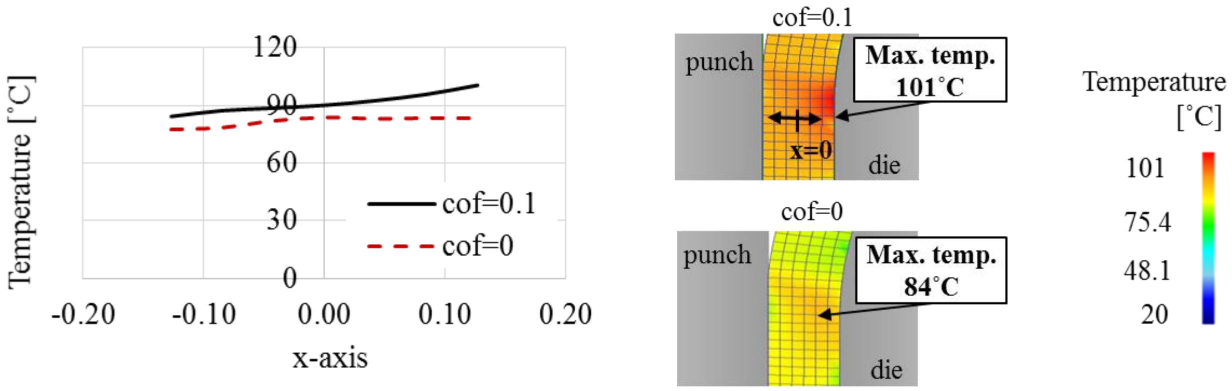


Figure 6. Temperature distribution along the cross section of blank for cof=0 and cof=0.1.

Secondly, initial assumption of HTC (See Table 3) was studied. Decreasing HTC from 40 kW/m^2·°C to 20 kW/m^2·°C [9] lowers blank/die interface temperature from 39 °C to 35 °C. In this study, HTC was kept 40 kW/m^2·°C.

Lastly, heat convection with the air was examined. The numerical analysis were conducted when heat convection with environment was 0.02 kW/m2·°C as suggested by the software used (i.e., DEFORM 2D) and compared with the results when air convection was neglected. It was found that the air cooling effect at the tool/workpiece interface is less than 1 % for the given parameters and therefore was not taken into account.

Analysis of Process Sequence. After the analysis of the first stage, the process sequence is simulated using the determined parameters; HTC=40 kW/m^2·°C, cof=0.1, HTC_air=0, production rate was 20 samples per minute (spm). Maximum contact temperature on the lower die is calculated for each stage and results are illustrated in Fig. 7. Blank/die interface temperature increases throughout the process sequence and reaches to 237˚C for the final drawing, Stage 6. However, it must be noted that the results show the trend for a single stroke and does not consider the multiple number of strokes. Additionally, the numerical model does not consider the cooling effect of the lubricant. As a result, the calculated temperatures are certainly higher than in production, where the cooling effect of the lubricant is present.

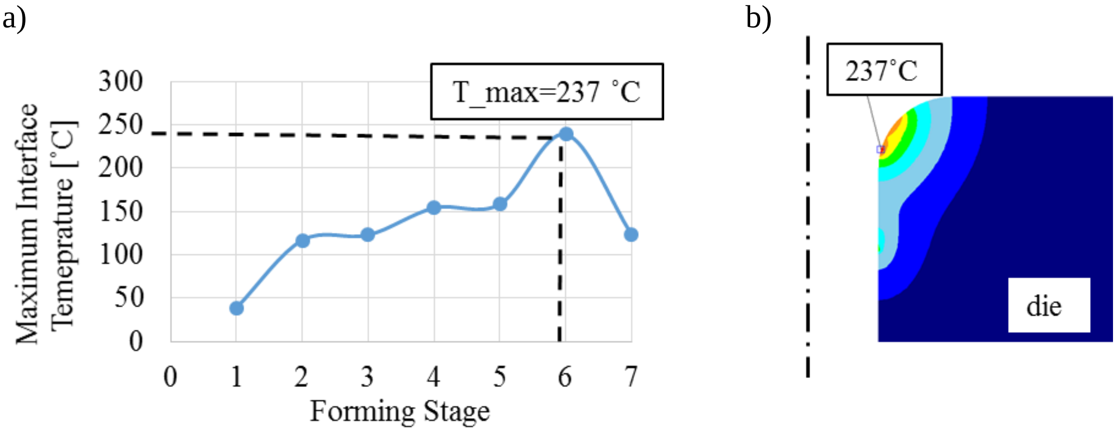
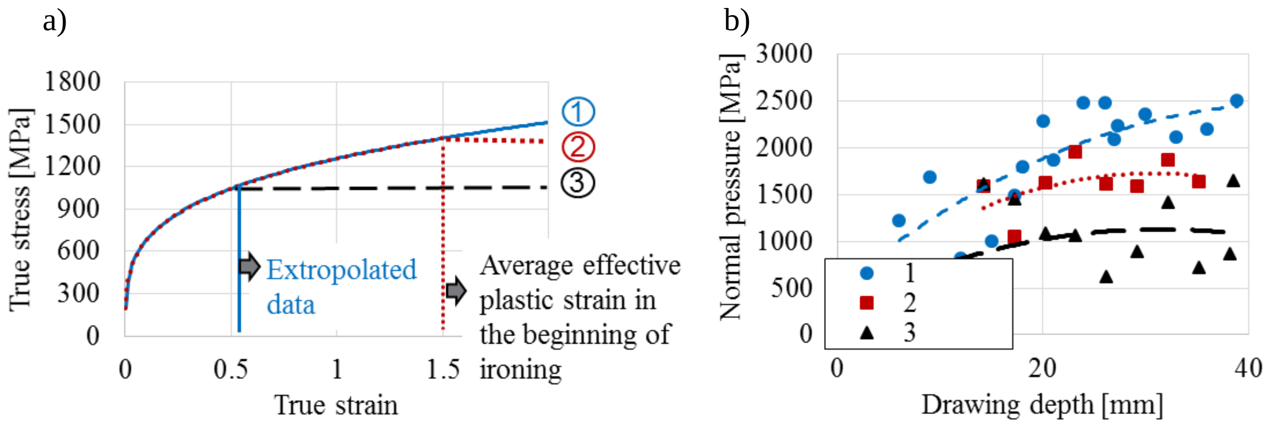


Figure 7. Illustration of a) calculated maximum contact die temperature with respect to drawing stages and b) temperature distribution in lower die at Stage 6.

In addition to the temperature analysis, contact pressure is another significant tribological parameter and therefore must be identified throughout the design. For the analysis of contact pressure, it was required to model the die as elastic material which results in very high CPU time, more than 10 hours. Due to that, only ironing operation, Stage 7 was examined in detail.



① $\bar{\sigma} = 1259\bar{\varepsilon}^{0.27}\ MPa$ ② $\bar{\varepsilon} < 1.5 \Rightarrow \bar{\sigma} = 1259\bar{\varepsilon}^{0.27}$; $\bar{\varepsilon} > 1.5 \Rightarrow \bar{\sigma} = 1405\ MPa$ ③ $\bar{\varepsilon} < 0.53 \Rightarrow \bar{\sigma} = 1259\bar{\varepsilon}^{0.27}$; $\bar{\varepsilon} > 0.53 \Rightarrow \bar{\sigma} = 1073\ MPa$

Figure 8. a) Various stress-strain curves used for numerical analysis of ironing stage and b) corresponding contact pressure as a function of drawing depth (Numbers 1, 2 and 3 correspond to various flow stress curve cases).

The contact pressure analysis shows that the normal contact stress reaches up to 2500 MPa for the material stress-strain relation given initially (See Fig. 8, Case 1). The calculated values are just below the yield strength of the tool material Vanadis 4, which is 3000 MPa for HRC 65 [10].

Numerical simulations may be overestimating the normal pressure due to the material stress-strain curve. The experimental data was obtained for the strains up to 0.53. The flow stress curve for larger strains, used in ironing (Stage 7), was extrapolated as given in Fig. 8a. In order to analyse the effect of the stress-strain relation of the material on the contact pressure, the true stress-strain curve was altered. In Case 2, the material is assumed perfect plastic after the end of Stage 6. The average plastic strain after the 6th drawing was calculated as 1.5 and the true stress was taken as 1405 MPa for the strains larger than 1.5 (See Fig. 8a, dotted line). In Case 3, the material is assumed as perfect plastic for the entire extrapolated region (See Fig. 8a, dashed line). The corresponding contact pressures given in Fig. 8b show that the stress-strain curve of the material has significant impact on the normal pressure.

Summary

In this study, designing a progressive die sequence for forming axisymmetric sheet metal parts including the consideration of metal flow as well as temperatures and pressures were analysed. First part of the study showed that existing numerical results which were previously compared with experimental results can be reproduced. The analysis of the mechanical model for the entire process sequence to produce an axisymmetric precision sleeve took less than 10 hours using processor Intel CPU e5-2620 (2.4 GHz). Commercial FE software, DEFORM-2D is beneficial to design more robust progress sequence, especially when 'know-how' information is limited. Additional investigations were performed to suggest blank/die contact temperature and pressure window. The major conclusions drawn from the study are:

- Any future work related to the selection of lubricants for the production line must consider that the temperature increases above 230 ˚C, without the cooling effect of selected lubricant.
- The material stress-strain curve affects the contact pressure calculations considerably, especially in ironing where strains are relatively large. It is therefore significant to determine the actual stress-strain curve of the material experimentally for larger strains.
- In order to develop a general approach to design process sequence and implement FE calculations in practical terms, it is vital to have close collaboration with industry.

References

[1] Naranje, V., and Kumar, S., 2014, "A Knowledge Based System for Automated Design of Deep Drawing Die for Axisymmetric Parts," Expert Syst. Appl., **41**(4), pp. 1419–1431.

[2] Yeon, S. M., Choi, S., Lee, G. A., Park, S. J., Kim, J. H., and Song, J. H., 2013, "Multi-Stage Deep Drawing Process in Thin Sheet for the Sleeve Housing Component of a Spindle Motor in HDD," Appl. Mech. Mater., **389**, pp. 359–363.

[3] Esche, S. K., Khamitkar, S., Kinzel, G. L., and Altan, T., 1996, "Process and Die Design for Multi-Step Forming of Round Parts from Sheet Metal," J. Mater. Process. Technol., **59**, pp. 24–33.

[4] Cao, J., Li, S., Xia, Z. C., and Tang, S. C., 2001, "Analysis of an Axisymmetric Deep-Drawn Part Forming Using Reduced Forming Steps," J. Mater. Process. Technol., **117**, pp. 193–200.

[5] Altan, T., Jain, N., Shi, X., Ngaile, G., Pax, B., Harman, B., and Homan, G., 2006, "Progressive Die Sequence Design for Deep Drawing Round Cups Using Finite Element Analysis," J. Manuf. Sci. Eng., **128**, pp. 366–369.

[6] Lange, K., 1985, *Handbook of Metal Forming*, Society of Manufacturing Engineering, Dearborn, Michigan, USA, Chapter 20.

[7] Altan, T., and Tekkaya, A. E., 2012, *Sheet Metal Forming - Fundamentals*, pp. 111.

[8] Ceron, E., 2013, “New Tribo-Systems for Sheet Metal Forming of Advanced High Strength Steels and Stainless Steels,” Denmark Technical University.

[9] Altan, T., Ngaile, G., and Shen, G., 2005, *Cold and Hot Forging,* ASM International, Materials Park, OHIO, Chapter 6.

[10] Uddeholm, 2016, *Vanadis 4 Extra SuperClean*.