

DEFORMATION ABILITY OF SINGLE POINT INCREMENTAL FORMING FOR THERMO-PLASTIC COMPOSITE MATERIALS

Pham Van Trung, Vo Van Cuong, Le Khanh Dien, Nguyen Van Nang, Phan Dinh Tuan,
Nguyen Thanh Nam

DCSELAB, University of Technology, VNU-HCM

(Manuscript Received on October 21th, 2010, Manuscript Revised January 21st, 2011)

ABSTRACT: *The studies of theory and simulation for composite materials have not yet built model for analyzing the effects of temperature's influence on the deformation ability of these materials. The article focused on studying the relationship between temperatures and deformation ability of composite sheets in SPIF method, building and analyzing simulation model, determining the limited deformation angle according to the temperatures and building the chart of limited forming for the polypropylene (PP) glass fiber type E mat composite material.*

Keywords: *forming ability, PP thermoplastic mat composite, SPIF of composite.*

1. INTRODUCTION

Among the previous rare works, plasticity theory combining with experiments often focus only on steel materials, aluminum and some research on composite materials, polymer. The study of theory and simulation for composites, thermoplastics have established the influence model of tool size ratio, the size of depth step[1], "Incremental Forming for the thermoplastic glass materials"[2], "SPIF in processing polymer materials"[3]. While temperature has great influences on the forming ability of polymer, composite but studies have not yet built model for analyzing the effects of temperature on the ability of this material. The article focused on studying the effects of temperature parameters to the forming ability of composite plates, finding out the limited forming angle according to temperature. The material in research is

polypropylene (PP) glass fiber type E mat composite[4].

2. THE COMPATIBLE FIBER-REINFORCED COMPOSITE MATERIALS FOR SPIF

Fiber-reinforced composite materials can be divided into two main categories normally referred to as short fiber-reinforced materials and continuous fiber-reinforced materials. Continuous reinforced materials will often constitute a layered or laminated structure. The woven and continuous fiber styles are typically available in a variety of forms, being pre-impregnated with the given matrix (resin), dry, uni-directional tapes of various widths, plain weave, harness satins, braided, and stitched.

The short and long fibers are typically employed in compression moulding and sheet moulding operations. These come in the form of flakes, chips, and random mate (which can

also be made from a continuous fiber laid in random fashion until the desired thickness of the ply / laminate is achieved).

For these features, the short-fibers composite has the ability of deformation larger than the others. Therefore, it turns out that the short-fibers are the most suitable material when processing with SPIF method. This type of composite will be used in the experiment to engage the larger deformation of the material.

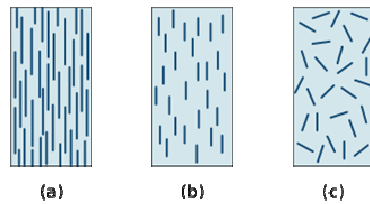


Fig. 1. Typologies of fibre-reinforced composite materials:

- a) continuous fibre-reinforced
- b) discontinuous aligned fibre-reinforced
- c) discontinuous random-oriented fibre-reinforced.

3. SETTING-UP SIMULATION MODEL

In the article, truncated cone is used as the object of analysis. To construct 3D parametric modeling, ProEngineer system is used. Toolpath data from NC code is translated into macros for preprocessing FEA. These macros defining the movement of tool use the constraints in some of load steps.

To build the forming limit diagram, a short-fiber composite type - composite sheet with thickness of 1mm - PP+20% glass fiber material was used. Components used in the calculation consist of: Young's elastic modulus $E=3.727\text{GPa}$, Poisson ratio 0.23, density

1268/m³. The processing temperature in simulation is done at the values 20, 60, 80, 100, 120, 140, 160, 165 degrees Celsius (0C), we used ABAQUS software to perform the analysis according to temperatures. Finally, using these results to build up the FLD (Forming Limit Diagram).

First, the simulation uses hidden FEA, encrypted ABAQUS and static analysis. The SPIF simulation process consumes a lot of time, so the higher-order approximation is used. Two most important parameters that the process time depending on are the number of nodes/elements in the model and the running segments of tool (number of load steps).

The initial conditions[1]: the model dimension is the circular plate with 80mm in diameter, 1mm in thickness. The tool radius is 5 mm, radius of gliding angle is 5 mm. Tool is moved along the circular toolpaths, down step is 0.5mm and the down distance total is 20mm.

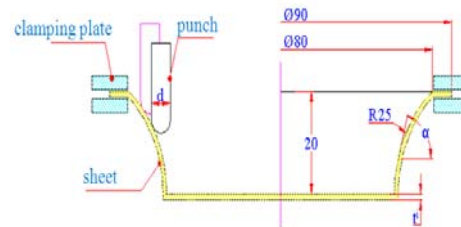


Fig. 2. The analysis model.

Analysing the deformation ability of sheet under environment of process temperature. At each different temperature, the deformation ability of sheets will be different, it means the angle α changes according to temperature. To assess the abilities occurring the torn phenomena and the detailed deformations on sheet processing by SPIF method, from the values of

forming limit angle α by the temperature, the forming limit diagram is built (describing the limits of deformation at which sheets can withstand without tearing).

3.1. Setting up CAD model

The forming ability of the sheet in the SPIF method is determined by the maximum forming angle α_{max} . The forming angle, where the sheet is not torned when being deformed, is the most important parameter, showing the limited forming ability of a specific material sheet.

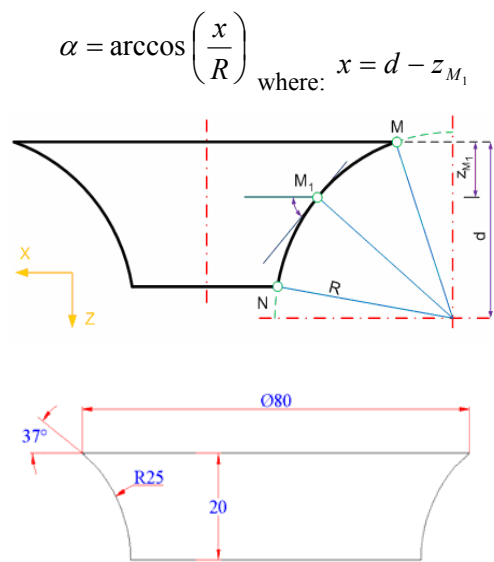


Fig. 3. The cone model in the experiments.

Since the forming ability of the composite material is quite good, the forming angle (α) will be surveyed in a range from 37 to 90 degree. In addition, due to the difficulties in choosing the computer time, the height of cone model will be limited at 20mm (Fig.3).

3.2. Building the CAM model

Performing the simulation process with the initial conditions as [1], in which authors had studied experiments with the optimal deformation parameters for thermoplastic as following: feed rate F 1000mm/min; spindle speed S 1200rpm; forming tool diameter d 10mm; vertical step (Z direction) 0.5mm.

3.3. Building the Analysis model in Abaqus software

- Starting Abaqus software



Fig. 4. The start screen of Abaqus software.

- Drawing the sheet model on the Sketch plane.

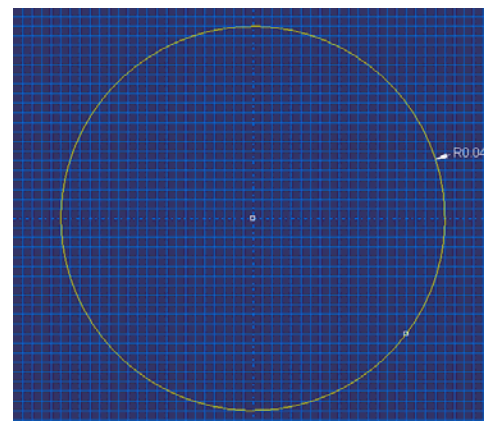


Fig. 5. Drawing sketch.

- Extruding the sheet with 80mm in diameter, 1mm in thickness.

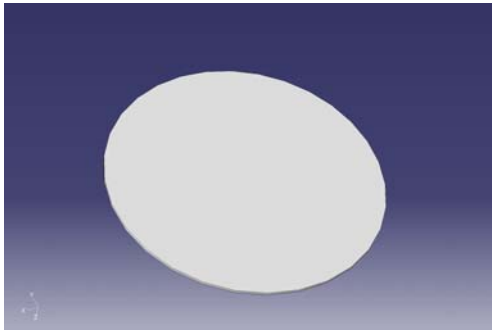


Fig. 6. The sheet model to analyse

- Creating the sketch of the punch with 10mm in diameter.

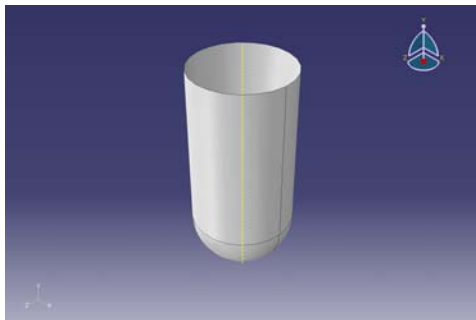


Fig. 7. The punch model in the experiments

- Choosing the composite material. Assigning the properties of material for the sheet: In section, creating the section composite. After that, on the Assign menu, choosing Assign → Section → clicking the sheet needing to assign the material properties → choosing Composite in the Assign Section table → OK.

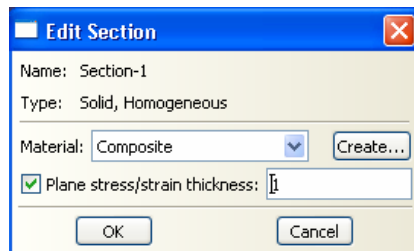


Fig. 8. The cone model in the experiments

- Assembling the analysis model: In Module, choosing Step. On Menu bar, choosing Step → Create, creating Step-1 to analyse according to Dynamic, Explicit. After that, the Edit step table occurs, entering the time of running NC file and Step to analyse.

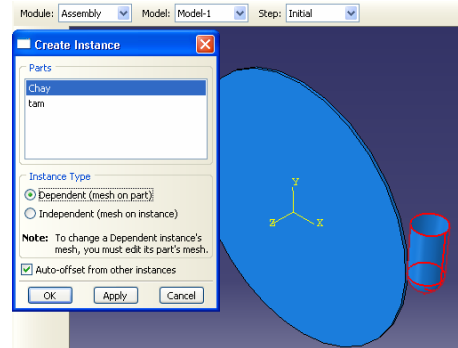


Fig.9. Assembling the analysis model

- Assigning the boundary conditions: Choosing the degrees of freedom and clicking at the boundary of the sheet.

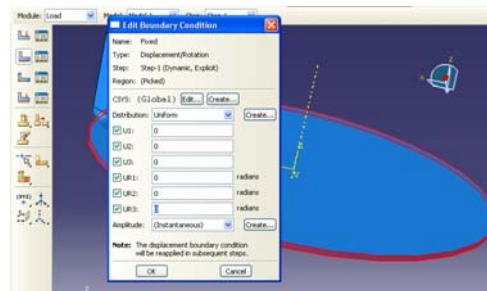


Fig.10. Assigning the boundary conditions.

- Assigning the moving conditions of tool according to NC file: Creating the Amplitudes x, y, z in turn. Such Amplitudes is the set of the coordinates of tool head by time, the tool will be moved to follow the curve approximating the coordinates.

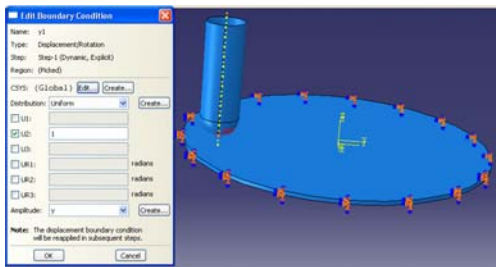


Fig.11. Assigning the moving conditions of tool

- Assigning the contacts between punch and sheet conditions: In the Module table, choosing Interaction. Next, clicking Interaction → Property → Create on the menu bar. The Edit Contact Property occurs, choosing the property of tangential contact. On the menu bar, clicking Interaction → Create to create the contact between the surface of punch and sheet. Clicking on the face of sheet, then choosing the face of punch in the direction of external contact (choosing Brown property)

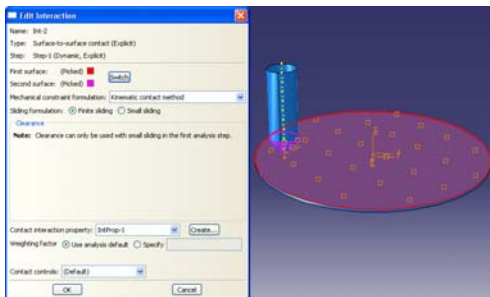


Fig.12. Assigning the contacts between punch and sheet conditions

- Meshing the analysis model: choosing Mesh in the Module table. On the menu bar choosing Mesh → Control & Element-Type, choosing punch and sheet needing to mesh. During the above steps, there occurs the property tables, we can choose the default values in these. On the menu bar, choosing Seed → Instance, the Global Seeds table occurs to assign the cell value of mesh.

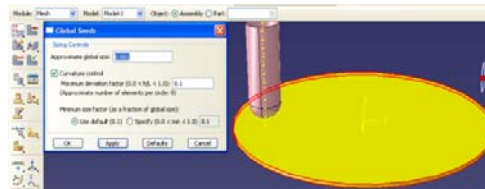


Fig.13. Meshing the analysis model

- Creating the analysis file: In the Module table, choosing Job. On the Menu Bar, choosing Job → Create → to name the analysis sample. After that, choosing Continue and OK. On the Menu Bar, choosing Job → Manager → Submit. The computer will perform to analyse.

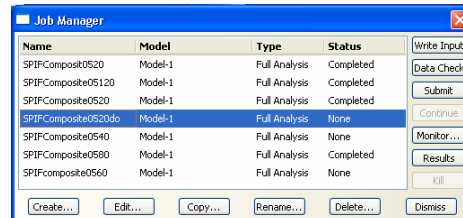


Fig.14. Creating the analysis file

3.4. Simulation

The process to simulate can be summarised in the following diagram:

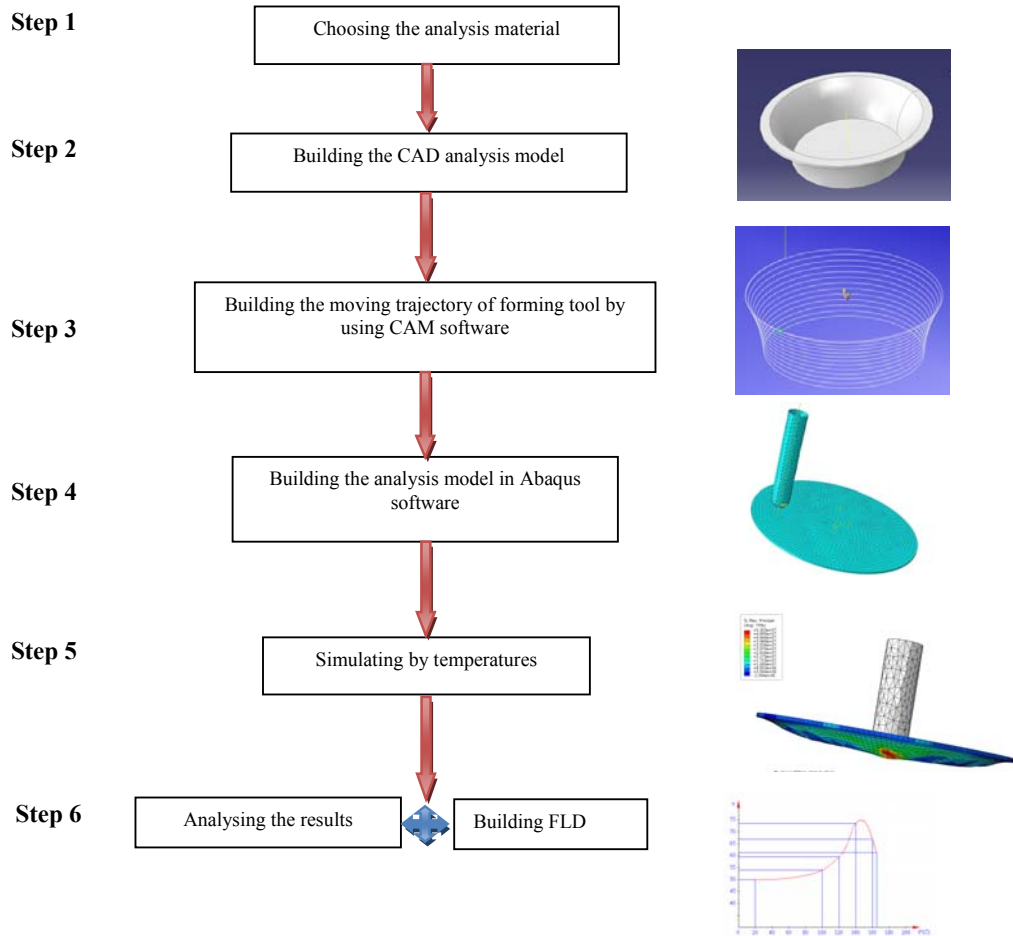


Fig. 15. Diagram of simulation steps

3.5. The simulation results

The simulation process using Abaqus software at the temperature values: 20, 60, 80, 100, 120, 140, 160, 165°C, we archived results in the following table:

Table 1. The simulation results.

Temperature (0C)	20	60	80	100	120	140	160	165
Limited forming angle (degrees)	50	51	52,7	53	59.6	74	67.2	61,5

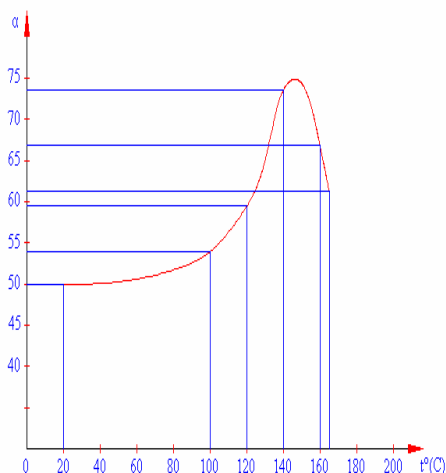


Fig. 16. Forming Limit Diagram by temperatures

3.6. Analysing the results

From the simulation results above, we recognised the limited forming angle changed considerably in the temperature range from 130 to 1650C. In the temperature range between 20

and 600C, the forming angle is almost unchanged. The maximum forming angle α_{max} is about 75 degree when the temperature is between 140 to 1600C (Fig.16).

4. CONCLUSION

The paper has studied the effects of temperature influence to the forming ability of the composite sheet in the SPIF method: building and analysing with simulation model, determining limited deformation angle (α_{max}) according to the temperatures and building the chart of limited forming of the polypropylene (PP) mat composite material, fiberglass-type E. From the temperature range 140 to 1600C, the limited forming angle is quite good ($\alpha_{max}=75$ degree). So, the result can be used to perform the SPIF process for the composite sheet in such temperature range.

NGHIÊN CỨU KHẢ NĂNG BIẾN DẠNG CỦA TẤM COMPOSITE NỀN NHỰA NHIỆT DẪO KHI GIA CÔNG BẰNG SPIF

Phạm Văn Trung, Võ Văn Cương, Lê Khánh Điền, Nguyễn Văn Nang, Phan Đình Tuấn,
Nguyễn Thanh Nam

DCSELAB, Trường Đại học Bách Khoa, ĐHQG-HCM

TÓM TẮT: Các nghiên cứu về lý thuyết và mô phỏng đối với vật liệu composite chưa xây dựng được mô hình phân tích ảnh hưởng của nhiệt độ đến khả năng tạo hình của các loại vật liệu. Bài báo tập trung nghiên cứu ảnh hưởng của thông số nhiệt độ đến khả năng tạo hình tấm composite trong phương pháp tạo hình cục bộ liên tục đơn điểm, xây dựng mô hình phân tử hữu hạn, phân tích với mô hình giả lập, xác định góc biến dạng giới hạn tạo hình theo các nhiệt độ và xây dựng được biểu đồ giới hạn tạo hình vật liệu composite nền nhựa polypropylen (PP) sợi thủy tinh loại E.

Từ khóa: forming ability, PP thermoplastic mat composite, SPIF of composite.

REFERENCES

- [1]. V.S. Le, T.N. Nguyen. K.D. Le, A. Ghiotti, G. Lucchetta., *Incremental Sheet Forming of Glass Mat Thermoplastics*. Proceedings of the Polymer Processing Society 24th Annual Meeting ~ PPS ~ Salerno (Italy).(2008).
- [2]. V. S. Le, A. Ghiotti, G. Lucchetta., *Preliminary Studies on SPIF for Thermoplastic Materials*. In: International Journal of material.(2008).
- [3]. P.A.F. Martins, L. Kwiatkowski, V. Franzen, A.E. Tekkaya and M. Kleiner., *SPIF in processing polymer material*; Journal: CIRP Annals - Manufacturing Technology Volume 58, Issue 1, (2009).
- [4]. Amirhossein Esfandiari, *Mechanical properties of PP/Jute and Glass Fibers Composite: The Statistical Investigation*, (2007).