

INJECTION MOLDING AND STRUCTURAL ANALYSIS IN METAL TO PLASTIC CONVERSION OF BOLTED FLANGE JOINT BY CAE

MARIÁN BLAŠKO, VIKTOR TITTEL*, ANTONÍN NÁPLAVA, MICHAL ONDRUŠKA

Slovak University of Technology in Bratislava, Faculty of Materials Science and Technology in Trnava, Institute of Production Technologies, J. Bottu 25, 917 24 Trnava, Slovakia

* corresponding author: viktor.tittel@stuba.sk

ABSTRACT. This contribution deals with replacing of metals part by plastics products. There are several benefits of this application — minimize part cost, corrosion resistance, integrating more components into one part etc. Material selection depending on the design of plastic part. It is necessary has to withstand the same load as metal part. To fulfill this requirement solve fiber reinforced plastics. Also it is convenient to substitute wall sections with ribbed structure. Mechanical properties this part could be important affected by fiber orientation. Results of fiber orientation can be used in stress analysis for better prediction to mechanical load. This analysis is performed in this study on bolted flange joint.

KEYWORDS: CAE, fiber reinforced plastics, injection molding simulation, metal to plastic conversion.

1. INTRODUCTION

Many metal parts in various applications are being replaced by plastic parts. There are several reasons for that depending on actual application — minimize part cost, enhance corrosion resistance, integrating more components into one single part etc. [1–3]. Most important steps in metal to plastic conversion are material selection and plastic part design. Plastic part has to withstand the same load as metal part. To fulfill this requirement fiber reinforced engineering plastics are often used. Mechanical properties of fiber reinforced part are highly affected by fiber orientation as a result of flow. However fiber reinforced materials are often treated as isotropic materials what could lead to potential problems with final dimensions, warpage and/or poor mechanical properties in highly stressed areas of part. Assuming fiber reinforced plastic part as isotropic is convenient in preliminary design phase and preliminary stress analysis to find out where are the critical most demanding areas in terms of stress. These areas should be redesigned to meet the stress requirements, however a reduced stiffness and strength should be considered in this phase. Injection molding simulation follows after the preliminary stress analysis [4]. For parts with heavy wall thickness and with thickness variations it is necessary to use 3D injection molding simulation to capture all phenomena related to fiber orientation. Important considerations in injection molding analysis are manufacturability of the part — the part should be void free, it should comply with warpage limits, dimensions etc. And the fiber orientation in the part should comply with mechanical loading of part to get the best possible mechanical properties for given applications. Fiber orientation is mostly affected by gate location and plastic part design [4, 5]. Final step in CAE approach is stress analysis with anisotropic material properties resulting from fiber orientation.

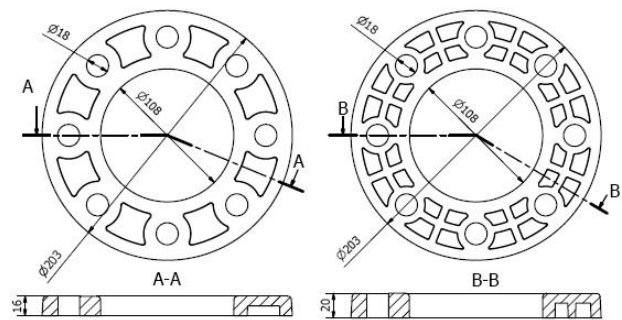


FIGURE 1. Ductile iron and PA66 GF30 flange.



FIGURE 2. Crack and crack detail.

Motivation for this study was an existing design of injection molded flange made of PA66 GF30. It is DN80 flange, pressure class PN10. This design was converted from metal — ductile iron flange. Both designs are shown in Fig. 1.

Flanges from PA66 GF30 are injection molded with cold runner system with one gate. PA66 GF30 flanges failed to pass the load test. During the load test, two fittings were coupled with flanges and bolted together. After tightening the bolts to prescribed torque 40 Nm, flanges cracked. Crack occurs mostly near the hole farthest from the gate. The location of the crack is in the thickest area of the part, wall thickness in this area is 17 mm. Crack location and crack detail are

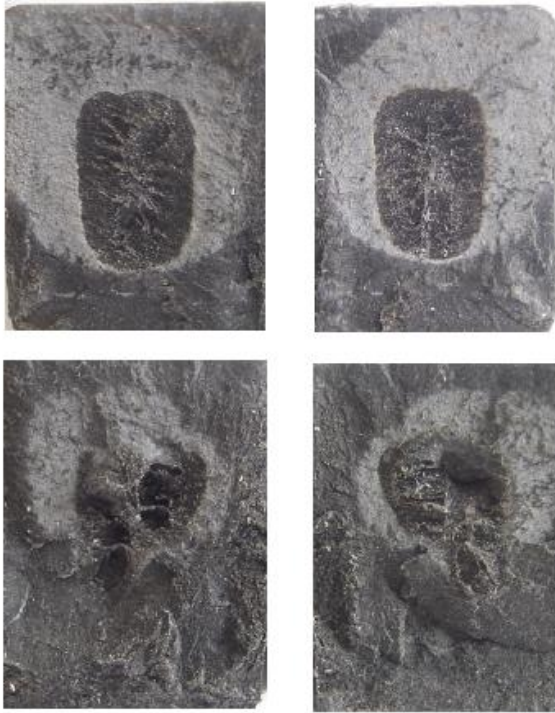


FIGURE 3. Details of fracture surfaces.

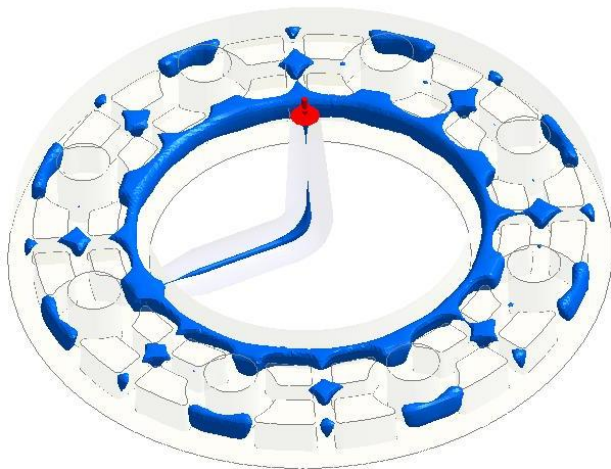


FIGURE 4. Melting core after packing phase.

shown in Fig. 2. Voids and a foam like structure are visible on fracture surfaces (Fig. 3). This implies that the packing phase was not sufficient.

Injection molding analysis was performed on actual flange model with corresponding feed system and injection molding parameters. Material in thickest locations is still far above melting point, while the cold runner system is already frozen (Fig. 4). Volumetric shrinkage can not be further compensated by adding new material, thus voids form in these locations. The largest void would be in the area with largest volumetric shrinkage, which is the thick section near the hole farthest from the gate as shown in Fig. 5 — this is also the area where material started to crack during

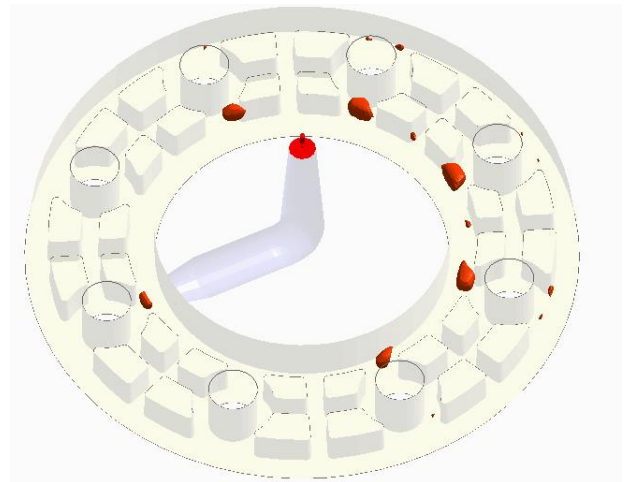


FIGURE 5. Areas with highest volumetric shrinkage.

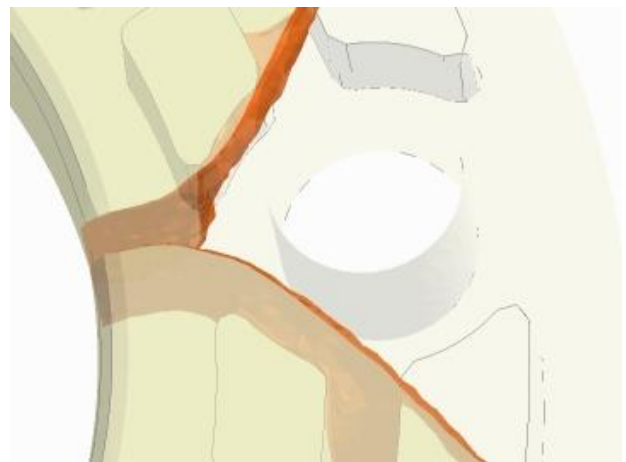


FIGURE 6. Weld line formation.

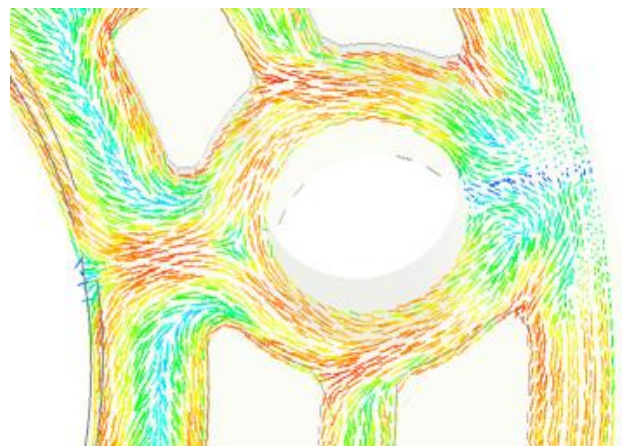


FIGURE 7. Weld line fiber orientation.

load test. However it is not only the void factor that contributed to the crack formation, but also the weld line in this location. Strength of PA66 GF30 decreases in weld lines about 15 to 17% [4]. Weld line formation and resulting fiber orientation are shown in Figures 6 and 7 respectively.

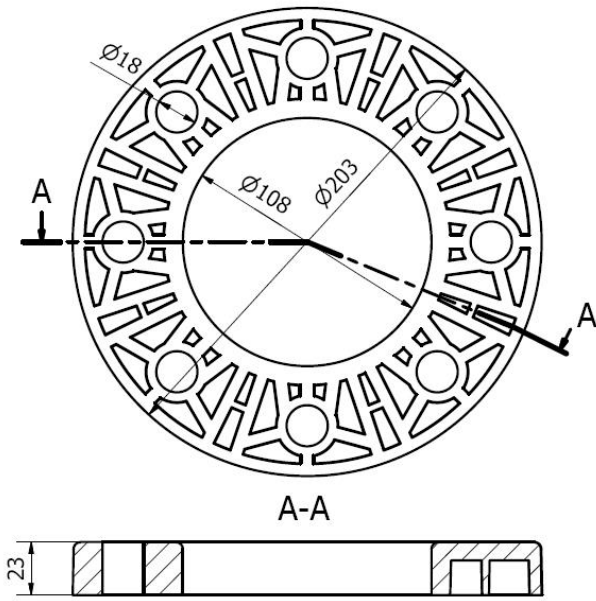


FIGURE 8. Flange redesign.

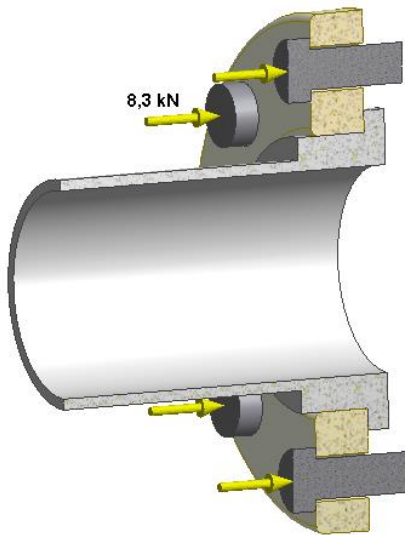


FIGURE 9. Setup for preliminary stress analysis.

2. DESIGN, MATERIALS AND ANALYSIS APPROACH

The goal in this study is to redesign the flange so it will meet the structural requirements (proposed design must withstand the load test) and also manufacturability requirements — proposed design must be void free after injection molding. Following CAE approach is used in this case study:

- redesign the flange,
- preliminary stress analysis of new design, design corrections, material considerations,
- injection molding analysis,
- stress analysis with consideration of fiber orientation.

PA66 GF50	Data sheet	Reduced
Tensile Strength	245 MPa	208 MPa
Tensile Elongation	3.0 %	
Tensile Modulus	16000 MPa	13600 MPa
Flexural Strength	360 MPa	306 MPa
Flexural Modulus	15000 MPa	12750 MPa

TABLE 1. Mechanical properties of PA66 GF50.

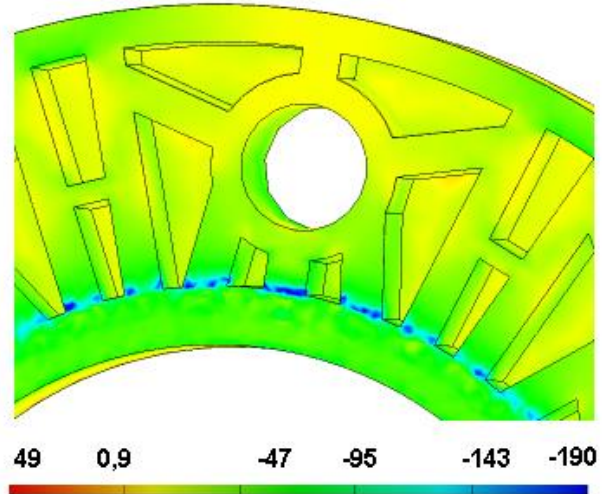


FIGURE 10. Maximum principal stress.

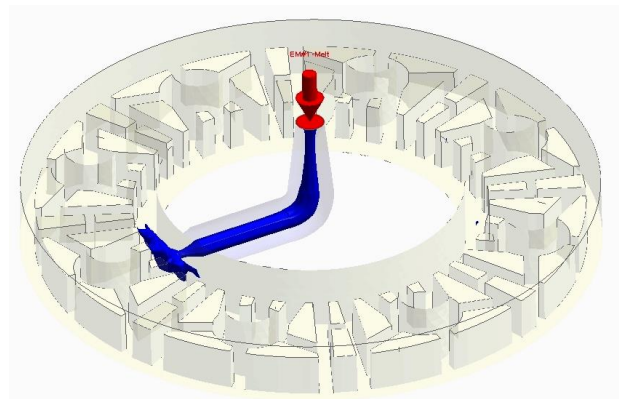


FIGURE 11. Melting core at the end of packing.

New proposed design of flange is shown in Fig. 8. Thick sections were cored out and more stiffening ribs were added to support the structure. Preliminary stress analysis setup is shown in Fig. 9. Clamping force 8.3 kN was applied to each bolt, which corresponds to the tightening torque of 40 Nm.

3. RESULTS

Preliminary stress analysis was first performed with PA66 GF30, however stress levels were high for this material, so PA66 GF50 was selected instead. Reduced mechanical properties were assumed according to Tab.1. by the calculation on basis software Moldex 3D user manual, version R20. Since PA66 GF50 has a brittle behavior when loading at room temperatures



FIGURE 12. Fiber orientation.

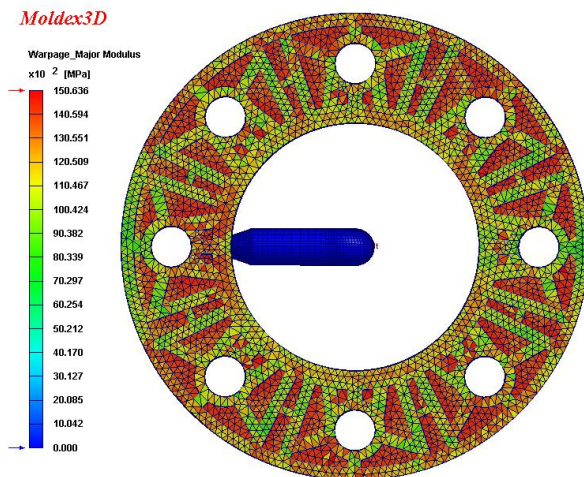


FIGURE 13. Major modulus.

a maximum normal stress theory can be assumed as a failure theory — however only in a “isotropic” analysis. Maximum principal stress in preliminary stress analysis reached 190 MPa as shown in Fig. 10, and it is in the locations where flange “bends” over fitting. Maximum deflection in z direction was 0.5 mm. These results seem to be good enough to go into injection molding simulation.

Injection molding analysis shown that new flange design can be molded without voids. Melting core at the end of packing is shown in Fig. 11. Highest volumetric shrinkage is 4.2% and it is near the gate. Fiber orientation is shown in Figures 12 and 13 shows resulting mechanical properties (major modulus) of flange based on fiber orientation. It is clear from this result that very low stiffness was achieved in stiffening ribs, only about 8000 MPa, a half of the data sheet modulus.

Material properties resulting from injection molding analysis were mapped onto structural mesh from Ansys (solid186). Resulting mapped mesh is “assembled” from elements with different material properties — according to orientation of fibers. Static structural analysis was performed in Ansys with the same loads as in preliminary “isotropic” analysis. Third principal stress result plot is shown in Fig. 14.

Results from “isotropic” and anisotropic stress anal-

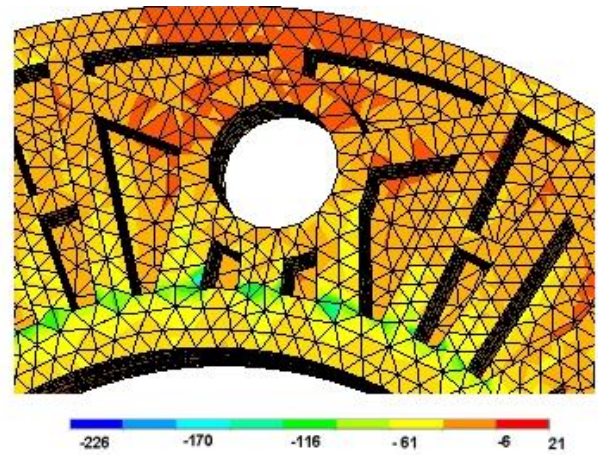


FIGURE 14. Third principal stress result plot.

Analysis	isotropic	anisotropic
1st principal	-97/102	-53/122
3rd principal	49/-190	21/-226
Von Mises	130	210
Displacement z [mm]	-0.5	-0.65

TABLE 2. Summary of results from stress analysis. The stress values are in MPa.

ysis are compared in Tab. 2. Further research is needed for clear statement on failure of this part. Also the part design and gating options have to be reviewed, since the highly stressed locations have poor fiber orientation in relation to loading of this part.

4. CONCLUSION

In this contribution CAE approach for designing a reinforced plastic part was presented. Mechanical properties of fiber reinforced parts are strongly influenced by fiber orientation resulting from injection molding process. Orientation is mainly affected by gating of the part and part design itself. “Isotropic” approach is not sufficient for predicting part behavior under load. It can be convenient in preliminary design to find out stress requirements a then redesign the part accordingly or select different material. Injection molding simulation is vital to avoid defects in molded part such as voids. And in case of fiber reinforced materials to analyze the resulting fiber orientation and get anisotropic material properties. Anisotropic material properties from injection molding simulation can be mapped onto structural mesh and then stress analysis of molded part can be performed. Anisotropic stress analysis gives better insight on how the part will perform under load, what deflections can be expected. However, since there is different orientation, it is difficult to tell what are the allowable stress levels and thus whether the part will fail or not. The CAE analysis were used for redesigning new type of injection molding tools and practical verification and mechanical testing of plastics part will be follow.

REFERENCES

- [1] G. Pötsch, W. Michaeli, Injection molding — An introduction, Hanser/Gardner, 1995.
- [2] J. Beaumont, Runner and Gating Design Handbook — Tools for Successful Injection Molding, 2nd Edition, Hanser/Gardner, 2007.
- [3] E. Campo, The Complete Part Design Handbook — For Injection Molding of Thermoplastics, Hanser/Gardner, 2006.
- [4] Moldex 3D user manual, version R10.
- [5] A. Y. Peng, W. Yang, C. David, Seamless integration of injection molding and structure analysis tools, Antec proceedings, 2005.