THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE V MACHINE BUILDING TECHNOLOGY. ISSN 1221-4566
2003

OPTIMIZATION OF THE SINGLE-CAVITY MOULD ACTIVE ZONE DIMENSIONING, FOR PLASTIC INJECTION-MOULDING, USING FINITE ELEMENT MODELS

Conf. dr. ing. Doina BOAZU,
Prof. dr. ing. Cătălin FETECĂU
"Dunărea de Jos" University of Galaţi

ABSTRACT

Knowing the injection moulding process requires, apart from knowing the properties of the materials involved and the details of the procedure itself, also, knowing the dependencies between the process parameters. The mould temperature is one of the most important factors in this process.

The paper presents an analysis of the thermal transfer taking place between the plastic material and the mould, using finite element models and considering that the cylindrical mould is a thick-walled tube.

1. Introduction

Given the actual continuous plastic goods development and diversification, the manufacturers are facing new challenges concerning the improvement of the quality of their products. This can be achieved through design optimization and proper construction of the injection moulds, which are decisive factors in manufacturing good quality thermoplastic products [1], [4]. The wide variety of plastic injected parts has lead to elaboration of specific constructive and technological solutions, both in designing and in execution of plastic injection moulds. These are, generally, very expensive and delicate tools, made of special steel and manufactured with precision machine tools, requiring very precise adjustments. That is why it is essential to optimize their design process, using finite element modeling.

Handling the injection moulding process requires, apart from knowing the properties of the materials involved and the details of the procedure itself, also, knowing the dependencies between the process parameters. The mould temperature is one of the most important factors in this process. The mould temperature must be chosen according with two possible objectives [3], [5], [6]:

-The technical quality of the injected part, as function of the temperature distribution uniformity and of the value of the mould temperature;

-The economy of the injecting cycle, due to the speed of mould temperature evacuation.

Next, we will analyze the thermal transfer between the plastic material and the mould, using finite element models, considering the cylindrical single-cavity mould as being a thick-walled tube.

2. The analysis, using finite element models, of the influence of the thermal transfer and injection pressure over the plane state of tensions within the mould

The following section presents the analysis, using finite element models, of the thermal transfer and injection pressure influence over the plane state of tensions within the single-cavity mould (figure 1), which is used to create the element presented in figure 2.

The parametrical finite element model, for the single-cavity mould, is represented in figure 3.

PLANE2D elements, with thickness of 20mm, had been used, in plane stress. The given parameter is the exterior radius R (fig. 3) of the cylindrical cavity without cap (3) in figure 1. The cylindrical mould is made of an alloy steel (material 1 in figure 3) and is mounted (forming a sliding fit) into a plate made of normal steel (material 2 in figure 3).

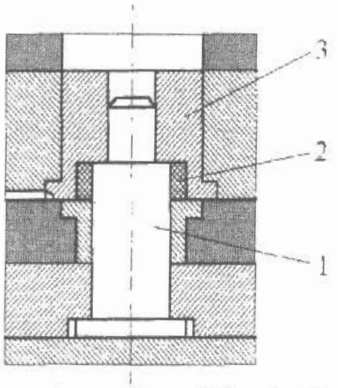

Fig. 1. The active region of the single-cavity mould used to obtain the "bearing" element: 1 - punch; 2 - the "bearing" element; 3 - cylindrical cavity without cap.

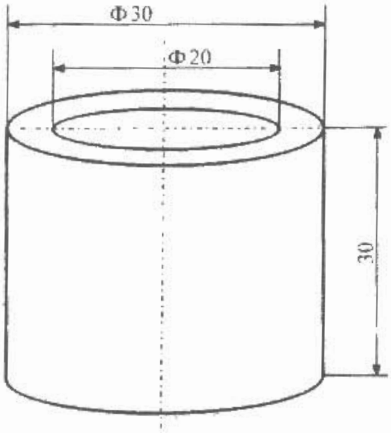

Fig. 2. The bearing element

In table 1 are shown the properties of the materials required by the thermal and static analysis.

Table 1.

Properties of materials

Property	Material 1	Material 2
Longitudinal elasticity module, E [N/m^2]	$2\cdot10^{11}$	$2\cdot10^{11}$
Poisson constant, ν	2,8	2,8
Linear thermal dilatation coefficient, α [grad^{-1}]	$1{,}1\cdot10^{-5}$	$1{,}3\cdot10^{-5}$
Thermal conductivity, k [m°K]	18	50

From table 1 one may conclude that these materials conduct temperature differently and there are also differences about their linear thermal dilatation coefficient.

In direct contact with the plastic material, the cavity is filled, with a pressure p=120MPa and temperature T=240°C. These loading conditions correspond to injecting common use *polystyrene*. On the exterior of the cavity, the temperature is supposed to be 50°C.

The temperature is not constantly distributed along the wall thickness, and, as a result, there is a supplementary stress, witch adds to the one caused by the pression on the inside.

The temperature distribution along the wall thickness is considered to be logarithmic.

$$T(r)=\frac{T_a}{ln\left(\frac{b}{a}\right)}ln\left(\frac{b}{r}\right). \tag{1}$$

Where:

r is the current radius of the thick wall tube;

a - interior radius;

b - exterior radius.

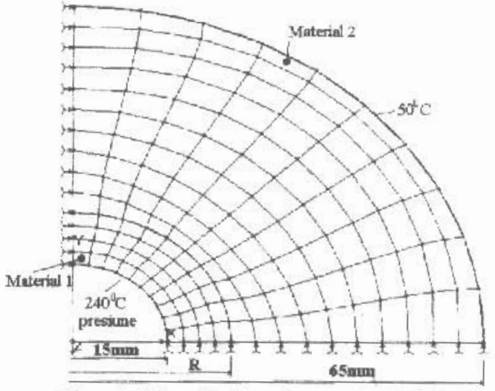

Fig. 3. The finite element model

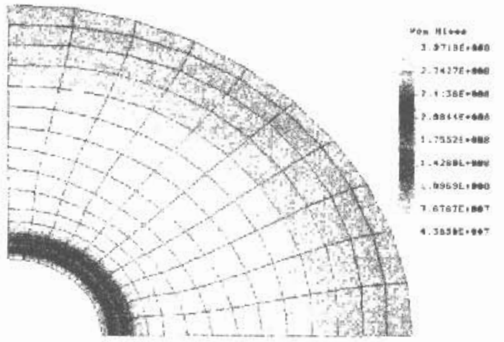

Fig. 4. The distribution of the Von Mises equivalent stress, caused by the differences in temperature

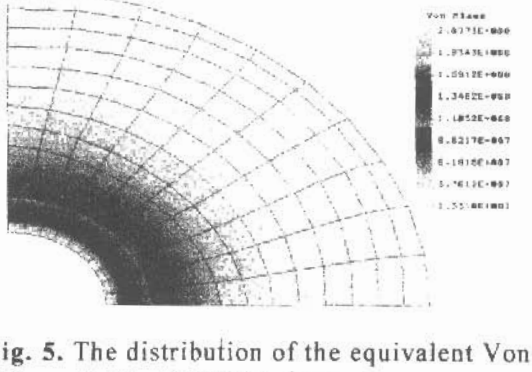

Fig. 5. The distribution of the equivalent Von Mises stress, caused by the interior pressure

The stress caused by the temperature distribution is bigger than the one caused by the interior pressure and, by acting together, they reduce each other's effects. This can be observed in figures 4 and 5, where for the parameter R, it was chosen a value R=25mm.

The optimization computations lead to the results in table 2.

Table 2.

Optimization results

Optimization parameters		Initial values and limits	Final value	Tolerances
Variable	R [m]	$20\cdot10^{-3}<25\cdot10^{-3}\leq25\cdot10^{-3}$	$21\cdot10^{-3}$	$1\cdot10^{-3}$
Goal function	Cube volume [mm^3]	2525132,75	13571,6	0,01
Restrictions	Von Mises stresses [MPa]	Mat 1 10<158≤250	152	4
		Mat 2 10<103<120	116	4

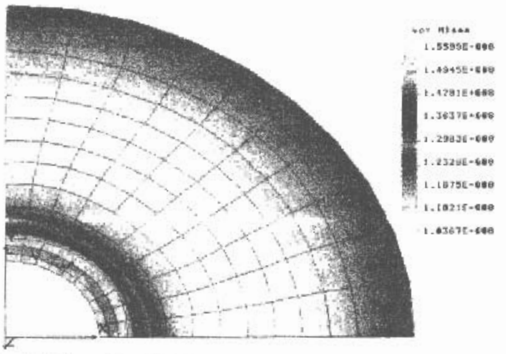

Fig. 6. The distribution of the equivalent Von Mises stress, caused by the interior pressure and the differences in temperature along the 6 mm wall thickness (injected material-polystyrene)

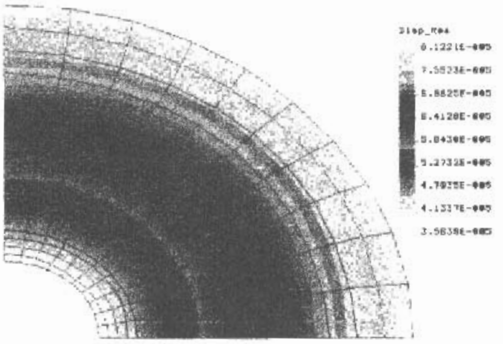

Fig. 7. The distribution of the displacements field caused by the interior pressure and the difference in temperature along the 6 mm wall thickness (injected material-polystyrene)

From table 2 it results a value of 6mm for the thickness of the mould sleeve (corresponding to $R=21\cdot10^{-3}$m), differing from the value of 7.5mm, which would result from the thick-walled tube theory (considering the mould not inserted into the base plate).

The final results, for the equivalent stresses distribution and for the displacements field, are given in figure 6 and figure 7, respectively.

It was considered, in the case of injecting *polypropylene*, that the maximum injecting pressure is 180MPa.

Choosing a mould wall thickness of 6 mm, for these new loading conditions, it is obtained a distribution of Von Mises stresses as shown in figure 7, in the case of polypropylene injection.

It results that the equivalent stresses increase with 3% in the case of injecting the material at a pressure of 180 MPa.

In strength calculation, in design procedures, it is considered, for simplicity, that the forming plate is either round or rectangular and in this plate there is nothing except the cavity of the mould, making abstraction of all the other bores. To be able to make the calculation, the forming plates with circular shapes are considered to be thick-walled tubes, under the influence of the internal pressure, resulting from the injecting process.

Generally, the interior diameter d of the forming plate is chosen according to constructive reasons, and the exterior diameter D is calculated (according to the maximum specific elongation theory) with relation [2], [9]:

$$D = d\sqrt{\frac{\sigma_a + 0{,}7p_i}{\sigma_a - 1{,}3p_i}} \tag{2}$$

where:

D - the exterior diameter;
d - the interior diameter;
σ_a - allowable stress;
p_i - interior injection pressure.

To obtain, through injection procedures, the element in figure 2, the injecting pressure is p_i= 1200daN/cm^2 and the cavity has an external diameter of D_e=45mm, an internal diameter of D_i=30mm and σ_a = 4500 daN/cm^2.

For dimensioning, it is taken into consideration the thermal gradient along the thickness of the bearing, during the injection process.

For finite element modeling, PLANE2D axial symmetric elements can be used. The model is presented in figure 8. The properties of this type of element are imported from the COSMOS/M program, and the properties of the material (alloy steel) are the ones in table 1.

Thermal loads (imposed temperature for the interior and exterior cylinders) and from the injection pressure, are given in figure 9.

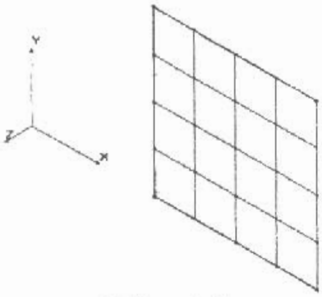

Fig. 8. PLANE2D axial symmetric finite element model

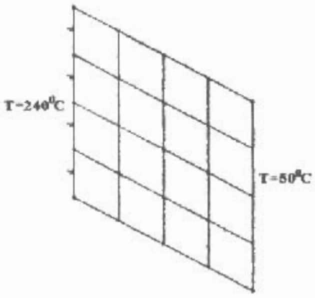

Fig. 9. Thermal and injection pressure loads

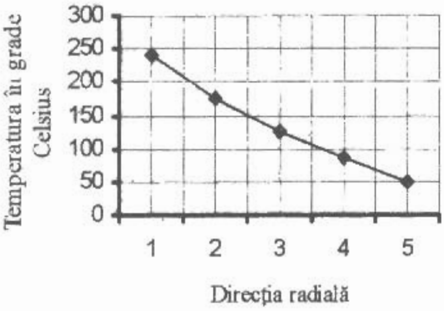

Fig. 10. Temperature variation along the thickness of the mould wall

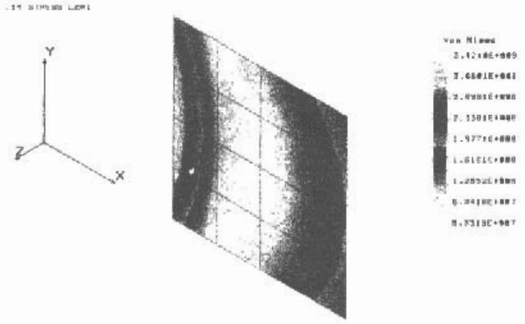

Fig. 11. Equivalent Von Mises stresses due to thermal loading

In figure 10 it is presented the temperature variation, along the thickness of the mould wall.

In figures 11...13 it is presented the variation of the equivalent Von Mises stresses, due to loading with interior pressure and thermal gradient, and in figure 14, the radial displacements due to loading with interior pressure and thermal gradient.

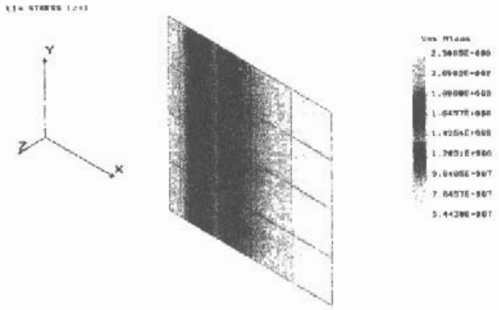

Fig. 12. Equivalent Von Mises stresses resulted from an internal pressure of 120MPa

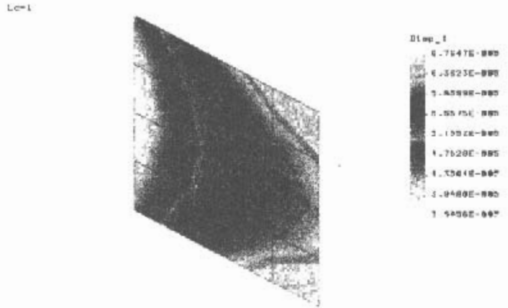

Fig. 13. Equivalent Von Mises stresses due to interior pressure and thermal gradient

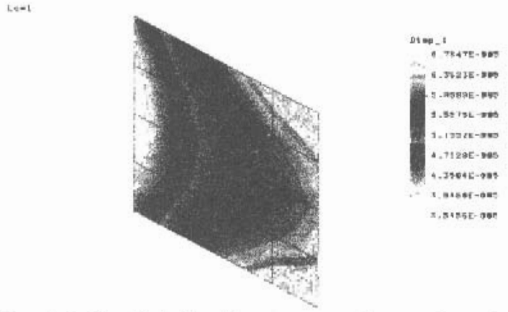

Fig. 14. Radial displacements due to interior pressure and thermal gradient loading

3. Conclusions

Dimensioning optimization for the mould active regions, for plastic materials injection, using finite element models, leads to getting more accurate results compared to those obtained using the thick-walled tubes theory.

The tests made on the models obtained from variation of the parameter R have evidentiated the fact that the stress on the exterior of the support plate has a concentration which increases with the thickness of the bearing; also, the displacements of the wall in contact with the plastic material tend to get smaller with the increase of the mould wall thickness, these displacements contributing to the tolerance of the exterior diameter of the resulted object.

The analysis of the influence of the thermal transfer and of the injection pressure on the plane stress within the mould, using PLANE2D axial symmetric finite element models in state of plane stress, allows a very accurate evaluation of the temperature variation along the thickness of the mould wall, of the variation of Von Mises equivalent stresses resulting from loading with interior pressure and thermal gradient and of the radial displacements due to the mould interior pressure and thermal gradient loading.

Bibliography

[1]. **Agassant, E. B.** - *La mise en forme de matieres plastiques. Technique et documentation, Paris, 1989.*

[2]. **Deutsch, I.** – *Rezistenţa materialelor, Editura Didactică şi Pedagogică, Bucureşti, 1979.*

[3]. **Feldman, D.** - *Tehnologia compuşilor macromoleculari, Eitura Tehnică, Bucureşti, 1974.*

[4]. **Fetecău, C., ş.a.** - *Prelucrarea maselor plastice prin injectare. Editat la OIDICM, Bucureşti, 1998.*

[5]. **Fetecău C., ş.a.** - *Polymer growth - Special and unconventional techniques. Symposium, October 14-15, Galaţi, pag. 212-218.*

[6]. **Ionescu M., ş.a.** - *Design of dies for injection manufactured plastic product, Editura Tehnică, Bucureşti, 1987.*

[7]. **Ponomariov, O C.** - *Calculul de rezistenţă în construcţia de maşini, Vol. II, Editura Tehnică, Bucureşti. 1964.*

[8]. **Tudose, R.** - *Equipment and processing in macromolecular compounds engineering. Editura Tehnică, Bucureşti, 1976*

[9]. **Zienkiewicz, O. C.** - *The Finite Element Method in Engineering Science, Mc Graw-Hill, london, 1976.*

[10]. *** - *Manual de utilizare COSMOS/M, Versiunea 2.5.*

Optimizarea dimensionării zonei active a matriţelor monocuib pentru injectarea maselor plastice folosind modele cu elemente finite

Rezumat

Varietatea deosebit de mare a produselor injectate din mase plastice a condus la elaborarea unor soluţii constructive şi tehnologice specifice atât în domeniul proiectării cât şi în cel al execuţiei matriţelor de injectat. Acestea sunt, în general, scule foarte scumpe şi pretenţioase care necesită, pentru confecţionare, oţeluri speciale, prelucrări cu maşini-unelte de precizie, ajustări foarte fine, fapt pentru care se acordă o atenţie deosebită optimizării proiectarii lor.

În cele lucrare se analizează tansferul termic între materialul plastic şi matriţă folosind modele cu elemente finite, considerând cuibul cilindric ca fiind un tub cu pereţii groşi.

Optimisation de la moisissure de cavité seule dimensioning de zone actif, pour injection du plastique - mouler, utiliser des modèles de l'élément finis

Résumé

Savoir le processus de la moulure injection exige, à part savoir les propriétés des matières impliqué et les détails de la procédure lui-même, aussi, qui savent les dépendances entre les paramètres du processus. La température de la moisissure est un des facteurs les plus importants dans ce processus.

Le papier présente une analyse du transfert thermique qui a lieu entre la matière plastique et la moisissure, en utilisant l'élément fini modèle et vu que la moisissure cylindrique est un tube épais muré.