

Analysis of a Stationary Platen of Injection Molding Machine

Prof. Parita Sheth
 Department of Mechanical Engineering
 Indus University
 Ahmedabad, Gujarat
 India

Prof. Amol Bagesar
 Department of Mechanical Engineering
 Indus University
 Ahmedabad, Gujarat
 India

Abstract- Injection molding is one of the most rapid and economical method of fabricating plastic materials in to the many simple, complex shapes & forms which is nowadays requirement of the highly growing plastic industry. The stationary platen made of spheroidal Graphite Iron of the Maxima-2300 tonnage injection molding machine is modeled by 3-D modeling software Pro/e 4.0 and analysis is done by Ansys 12.0 to check whether the obtained result is within permissible limit or not and than the obtained result is than justified logically.

Keywords- Injection Molding Machine, Stationary Platen of IMM, S G Iron

I. PROBLEM STATEMENT

To analyze a stationary platen of Maxima 2300 Injection Molding Machine (IMM) made of S G Iron, with the help of the software Ansys 12.0 and to check that whether the result obtained is within permissible limit or not.

II. BASIC OPERATIONS OF INJECTION MOLDING MACHINE

Raising the temperature of the plastic to a point where it will flow under pressure then allow the plastic to solidify in the mold, when the machine is closed. After opening of the mold to eject the plastic after keeping the material confined under pressure as the heat (which was added to the material to liquefy it) is removed to solidify the plastic and freeze it permanently in to the shape desired.

A. Force diagram of IMM

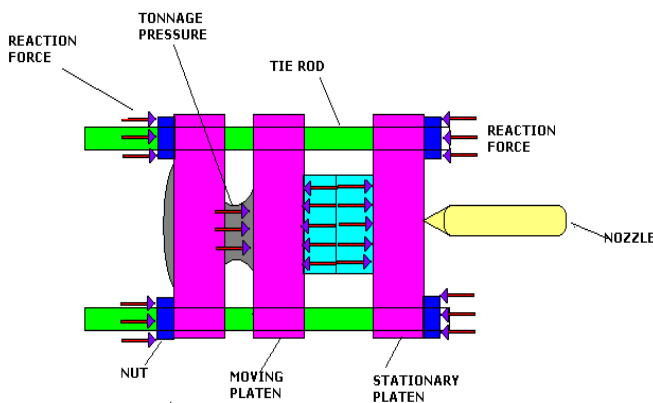


Fig. 1 Force Diagram of IMM

III. STATIONARY PLATEN OF MAXIMA 2300 IMM MODELLED IN PRO/ ENGINEER

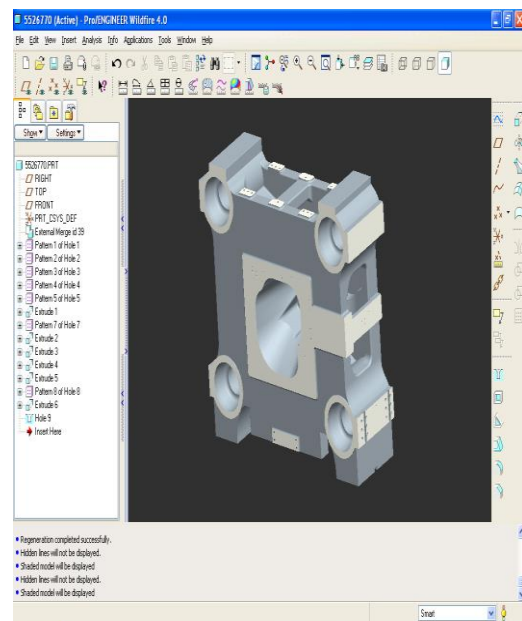


Fig. 2 Stationary Platen of Maxima 2300 Injection Molding Machine

IV. ANALYSIS OF STATIONARY PLATEN OF MAXIMA 2300 IMM

ANSYS is a general purpose finite element modeling package for numerically solving a wide variety of mechanical problems. These problems include: static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems.

TABLE1 Calculation sheet

	MAXIMA-2300		
DESCRIPTION	DATA	RESULT	UNIT
Force in Tonnes	2300	2300	Tonnes
Force in kgf	2300*1000	2300000	Kgf
Force in N	-	22563000	N
Force per Strain Rod	-	5640750	N
Centre Distance between Tie-Bar	Horizontal	2325	Mm
	Vertical	1925	Mm
Strain Road Diameter	-	325	Mm
Open Space	Horizontal	2000	Mm

	Vertical	1600	Mm
CALCULATION - MINIMUM MOLD AREA	Horizontal	2000	
	Vertical	1600	
	2/3 Horizontal	1333.33	Mm
	2/3 Vertical	1066.67	Mm
	Mold Area	H*V	1422222.2
Area of Bore	Dia of Bore	192	
Total Area	Mold Area- Area of Bore	1393269.30	
Pressure on Mold Area (P2)	-	16.19	N/mm ²
CALCULATION - AREA OF NUT			
Outer Dia of Nut	Do	640	Mm
Inner Dia of Nut	D1	510	Mm
Area of One Nut	-	117417.0254	mm ²
Pressure on one Nut (P1)		48.040	N/mm ²
S G IRON		81-117	Mpa

A. Ansys Steps

When the particular file is saved as pro/e form than select pro/e option or when the file is saved as .iges form than select iges option.

File > import > pro/e or iges
Or

To open from .anf file open > look in > open.

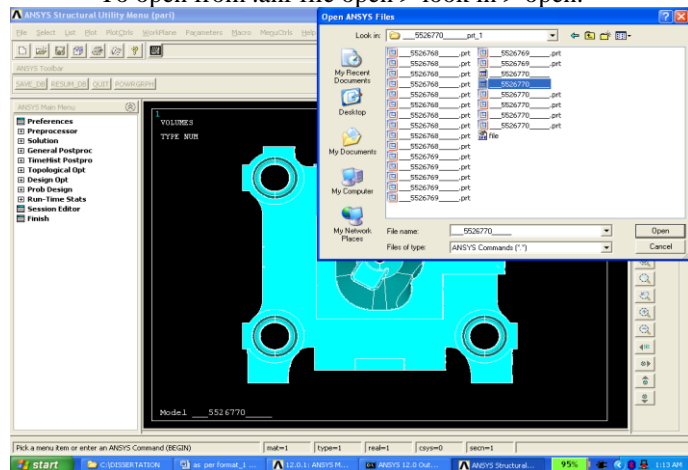


Fig.3 Open File in Ansys

B. Meshing of Volume

Main Menu > Preprocessor>Meshing>Mesh Tool>Size>15>volume> free> Mesh>Pick All >OK

Utility Menu > plot >volume or element

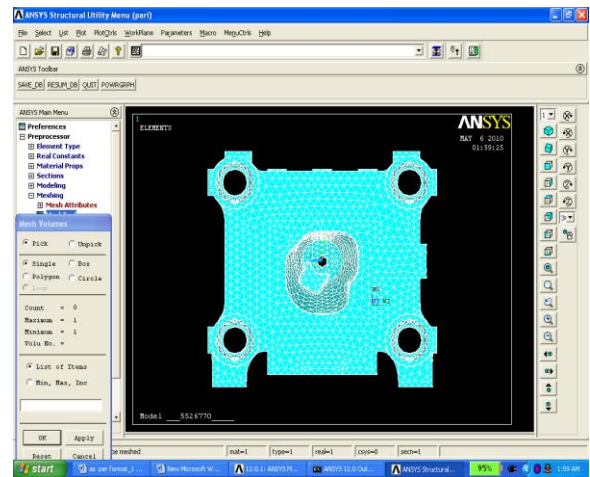


Fig.4 Result

Main Menu > General Postproc >Plot Results >Contour Plot >Nodal Solution> stress > 1st principal >ok

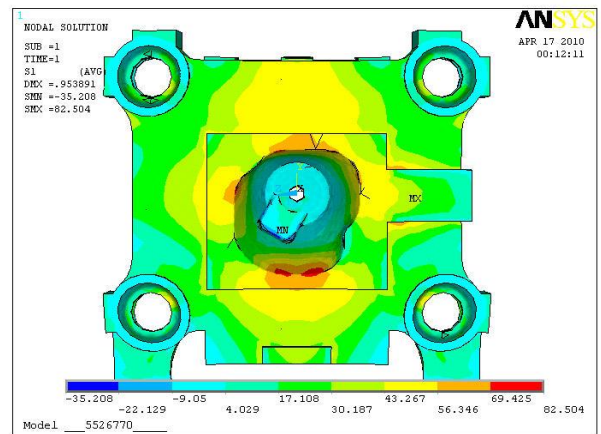


Fig. 5 Ansys Result

V. JUSTIFICATION FOR THE RESULT

The obtained result by Ansys is justified here by means of the force diagram and the front and rear view of the platen.

Whereas the other face towards nozzle is subjected to tensile stress which gives higher stress contours as shown in below figure6 (a).

As per the force diagram the mold mounting face is under compression while the other face towards nozzle is under tension; so the mold mounting face gives less stress contours (blue color)which is justified as it is subjected to compressive force only. Similarly the plate is bolted down so it shows higher stress near the leg as shown in figure 6 (b).

VI. CONCLUSION

The stationary platen of MAXIMA 2300 IMM is modeled and analyzed in this paper. After the Analysis of this platen it is found that the result obtained by Ansys is within permissible limits (<117 MPa) and the obtained result is justified logically too.

REFERENCES

- [1] Gallagher R. H.: 'Flat Plate Analysis', New Jersey ,Prentice-Hall, inc.
- [2] Timoshenko S. P., Woinowsky: 'Theory of Plates and Shells', New York, McGraw-Hill
- [3] Timoshenko S. P.: 'Theory of elastic stability', New York, McGraw-Hill:
- [4] Wojtaszak: 'The Calculation of Maximum Deflection, Moments, and Shear for Uniformly Loaded Rectangular Plate with Clamped Edge' J. Appl. Mech. Trans. ASME
- [5] Young: 'Analysis of clamped rectangular plates' J. Appl. Mech. 62

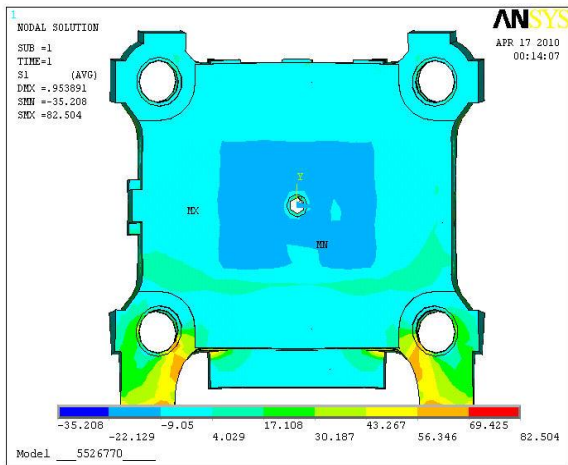


Fig:6(a)

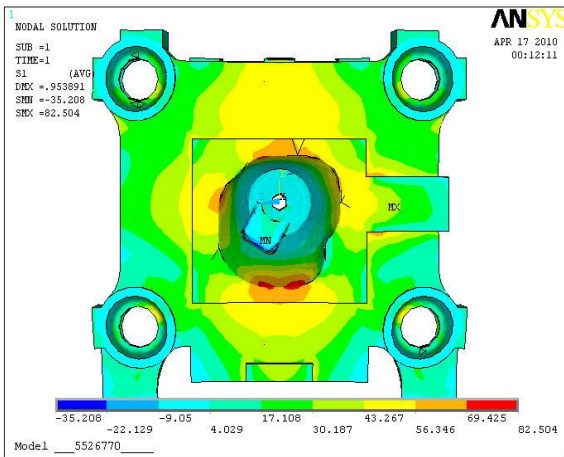


Fig:6(b)

-Fig.6 View of Stress Plot Minimum Mold Area