International Journal of Research in Engineering and Science (IJRES)
ISSN (Online): 2320-9364, ISSN (Print): 2320-9356
www.ijres.org Volume 10 Issue 7 ǁ July 2022 ǁ PP. 114-132

Optimization of Feed System Design in Three Plate Mold for Transparent Case of Electric Meter by Mould Flow Analysis

Md. Abdul jawad[1], Dr. R Nagaraja[2]
[1] *Student, Department of Tool Engineering,Government Tool Room and training Centre, Bengaluru*
[2] *Principal, Department of Tool Engineering, Government Tool Room and training Centre, Bengaluru*

Abstract- *This paper focuses on Optimization of feed system design in Three plate mould for transparent case of electric meter , That is mostly used in house hold appliances in electric meter and it is mounted as front transparent case in electric meter to read down meter readings.The model was reverse engineered using digital vernier calipers, a radius gauge, a height gauge and a micrometer.After that part was created in solidworks .The shape and size of part is sophisticated with wall thickness 3mm.This paper describes design of new feed system over existing feed system .In this paper current feed system which includes runner and gate will be redesigned.As existing feed system using single trapezoidal runner and single pin point gate .By designing double trapezoidal runner and two pin point gates and performing mould flow analysis to fill the part faster to reduce defects such as sink marks ,warpage and to reduce filling time, cooling time which leads to reduce cycle time which decreases production time,In first three cases using double trapezoidal runner placing gate at different location and choosing optimized gate location.In Fourth case changing runner design that is using semicircular or half round runner and comparing it defects, cycle time,filling time with newly designed double trapezoidal runner of first three cases*

Keywords:
Mould flow analysis Solidworks plastics,Optimized feed system design,Trapezoidal runner,Half round runner,pin point gate

---

Date of Submission: 24-06-2022 Date of acceptance: 05-07-2022

I. INTRODUCTION

Injection moulding is a method to produce plastic parts by injecting hot molten plastic material into a mould then cooling and solidifying upto a temperature where it becomes rigid ,The method is suitable for mass production of thermoplastic material products with complicated shapes, size and takes a large part in the area of plastic processing from househould products to engineered products we see in our daily life such as automotive interior trims,medical products such as injection,syrup bottles etc househould products food containers , buckets ,plates, tooth brush etc .the profile core and cavity determines component shape and size .In mould design three plate mould with pin point gate is used for auto degating for faster production ,cooling system is very

Essential it must be done uniformly to reduce surface defects.The proper manner of ejection should be choosen based on component profile .Pin ejection was used in this scenario because it is the best ejection system for flat components and easy to manufacture pins.

II. PROBLEM STATEMENT

Here a transparent case of electric meter having 96mm length,96mm width and 10 mm thick with suitable internal step wall thickness and 12 projections heighted above the surface with circular 4 holes is having defects which is not good for part quality and appearance so to avoid such defects by redesigning current feed system by taking two cases ,firstly choosing optimized gate location and then redesigned runner finally analyzing it defects and comparing it with existing feed system design

III. OBJECTIVE OF THE WORK

1.To analyze the existing feed system design and foresee the possible problem
2.To redesign the current feed system by placing gate at different location and using different runner and changing the dimensions of runner

3.To design and analyze the feed sytem in Four cases such as

Case 1. Existing single runner and single pin point gate
Case 2..Modified double Trapezoidal runner and two gates ,dimesions of sprue runner and gate same as existing feed system
Case 3. Modified double trapezoidal runner and two gates, reducing the distance of gate from centre to sprue
Case 4.Modified double half round runner with two pin point gates,
4.performing mould flow analysis of above four cases and selecting the optimized design with less surface defects such as warpage,sink marks, shrinkage and part filling with less filling time ,cooling time
5.Cooling system and sprue dimension remains same for the all Four cases.

IV. PART DETAILS

The component is made up of Polycarbonate which is amorphous in nature .Polycarbonates are strong,tough material and are optically transparent because of this it is used in front case of electric meter to read down the meter readings easily.Fig 1.1a) shows Actual part ,Fig.1.1 b) and Fig.1.1c) shows 3D Model and 2D draft of component designed in solidworks 2019 Software

Material:Polycarbonate
Part weight:30.51 grams
Density:1.2 g/cm^3
Volume:25638.55cm^3

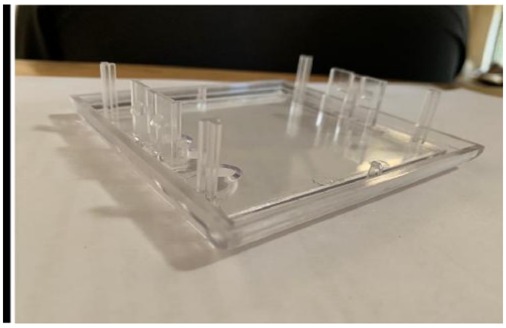

Fig. 1.1a) Actual Transparent case of an electric meter

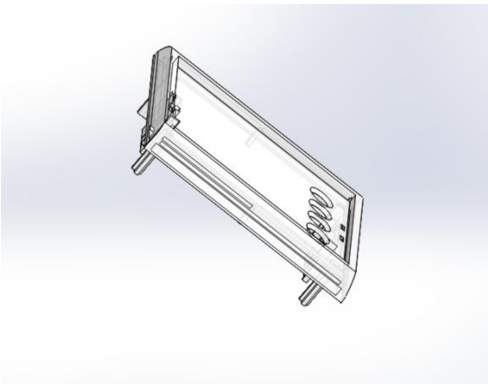

Figure 1.1b) 3D model of Plastic part

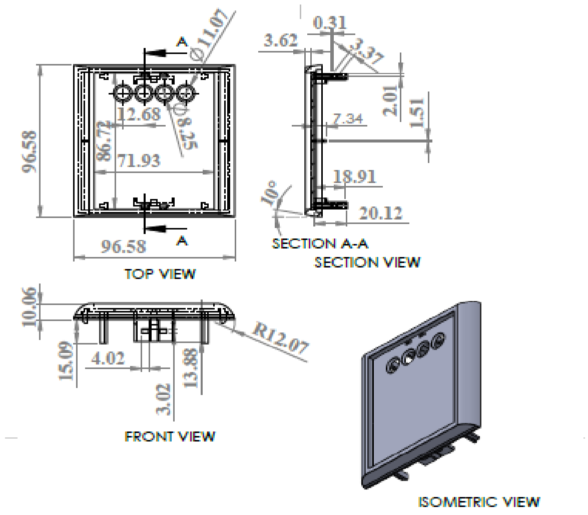

Fig. 1.1 c)2D model of part with 0.6% shrinkage added

V. MECHANICAL PROPERTIES OF POLYCARBONATE

Shear rate ,viscosity,density and temperature graphs

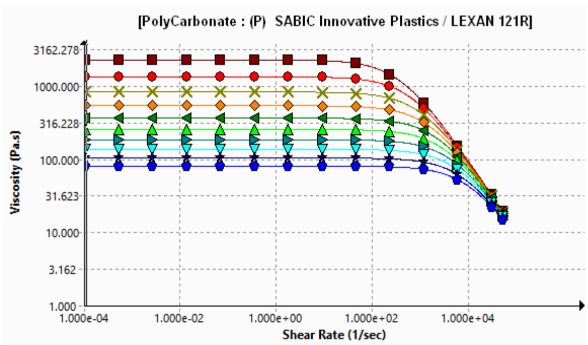

Figure 2 .Shear Rate(1/sec) vs viscosity(pa.s)

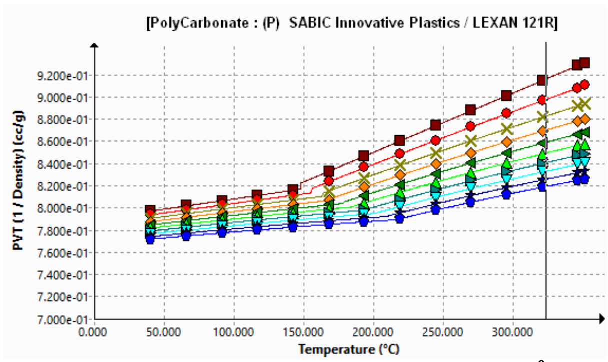

Figure 2.1a) Density (cc/g) vs Temperature (^{0}C)

5.1 Process settings

- Melt temperature:300(^{0}C)
- Mold temperature:100(^{0}C)
- Injection location:Centre of sprue
- Maximum Machine injection pressure:1250 bar
- Machine tonnage :80 Tonns

5.2 Mold tool design data

Mould set or Mould tool has been designed using standard Mould Base sytem of Gttc Standards

- Mold Material For core and cavity:H13 steel

- Top Plate:X-296mm Y-22mm Z-296mm
- Bottom Plate: X-296mm Y-22mm Z-296mm
- Core Plate: X- 246 mm Y-40mm Z-246 mm
- Cavity Plate:X-246 Y-50 mm Z-246mm
- Ejector Plate:X-170mm Y-12mm Z-246mm
- Ejector Back Plate:X-170mm Y-12mm Z- 246mm
- Spacer Block: X-36 mm Y-46mm Z-246mm

5.3 Material Data

Material	Polycarbonate
Mold Temperature Range	80-100^{0}C
Melt Temperature	280-320^{0}C
Ejection Temperature	175^{0}C
Glass transition Temperature	147^{0}C
Modulus of Elasticity	2350 MPa
Poisson's Ratio	0.38
Maximum shear stress	0.495 MPa
Maximum shear Rate	40000 1/S
Melt Flow rate	21 g/10 min

5.4 Coolant information

	Cooling channel core side	Cooling channel cavity side
Coolant used	Water	Water
Hose Diameter	Circular channel 8mm	Circular channel 8mm
Cooling cicuit	Rectangular cooling circuit/channel	Rectangular cooling circuit/channel
Flow rate	9.0 (lit/min)	9.0 (lit/min)

As shown from table coolant information and cooling circuit used is Rectangular fig.3)shows 2d draft of Cooling circuit

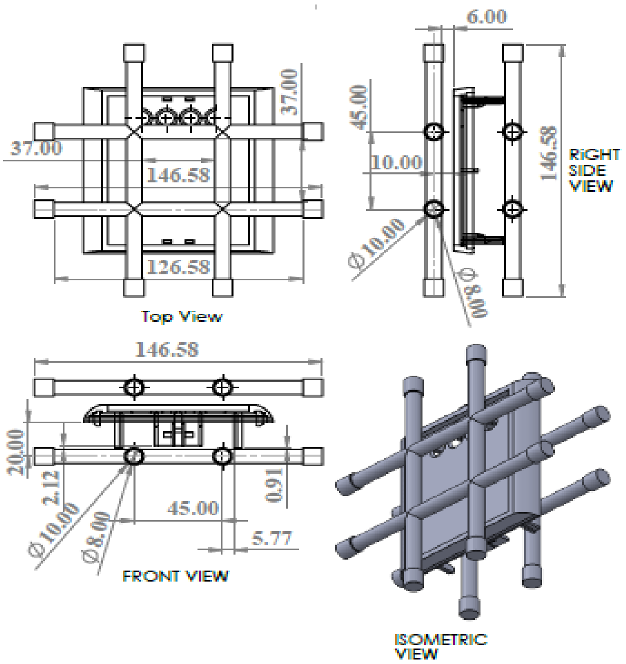

Fig.3)Cooling channels 2d draft

VI. MOULD FLOW ANALYSIS IN SOLIDWORKS

Using Mould flow analysis software we can analyze the part filling time,cooling time,overall cycle time and defects like shrinkage ,warpage,sink marks etc Before actual manufacturing of tool we can analyze the part and its parameters in CAE softwares like Nx mould flow analysis ,solidworks Mould flow analysis,Then performing analysis by changing feed system design and selecting optimized feed system

<u>Steps for Mould flow analysis in solidworks</u>

First open solidworks plastic and solidworks flow option by clicking on it

1.First creating mesh either solid mesh/shell mesh we have choosen shell mesh manual option then in that first step is to select cavity domain then selecting cooling channel domain

As shown from fig. 3.1a) Cavity ,Runner and gate selection

From fig.3.2 b) Cooling channel selection

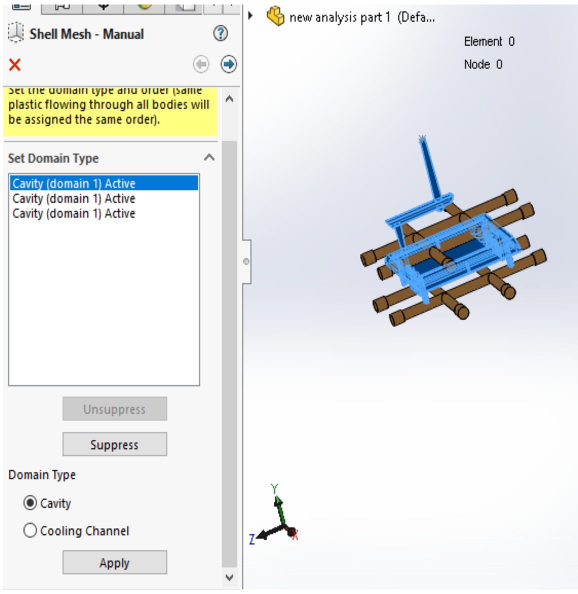

Fig.3.1 a) Cavity ,Runner and gate selection

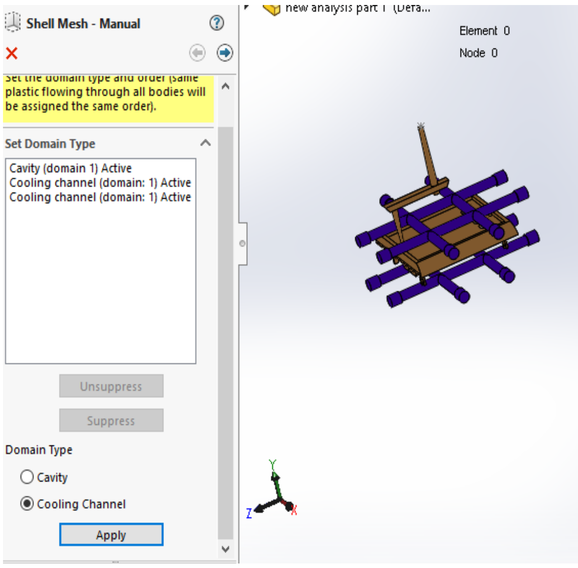

Fig. 3.1b) Cooling channels selection

Mesh geometry for Cavity,Runner is equilateral triangle with size 2mm

Mesh geometry for Cooling channel is equilateral triangle with size 3mm

Figure 3.1c) shows Final Meshed Component

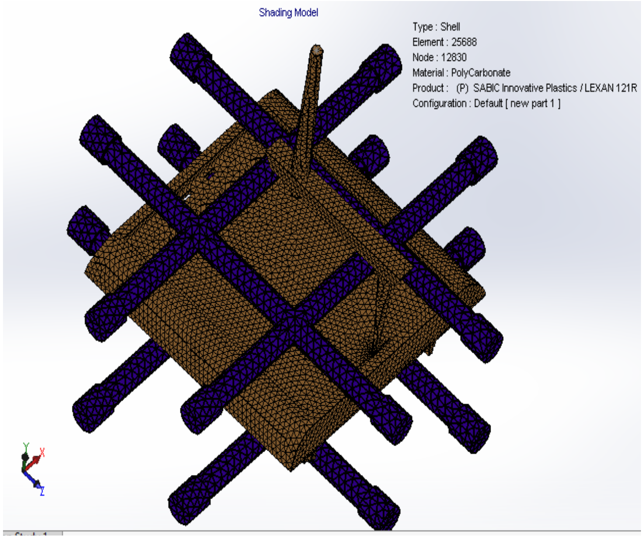

Figure 3.1c) Final Meshed component

2.Material selection

a. Polymer material selection from data base

- We have selected Polycarbonate Material
- Enter value such as Melt temperature -300^{0}C

b. Mould Material selection

- As runner and gates are machined in cavity itself select 154-H13 steel (Hot die steel) as mould Material

c. Coolant fluid selection

- Select coolant as water as it is standard and easy available

3. Applying Process parameter

a.Fill settings

- Mold temeperature-**100^{0}C**
- Melt temperature-**300^{0}C**
- Injection pressure limit-**57.2708 MPa** by calculation
- Clamping Force-**80 Tonns**

b.Warp parameters

- Ambient temperature-Select **30^{0}C**
- Gravity direction in Which our Molten plastic flows – **(-Y direction)**

c.**Coolant settings**

- Air Temperature-**30^{0}C**
- Minimum Coolant Temperature-**15^{0}C**
- Average Coolant flow-**9 lit/sec**
- Ejection Temperature-**175^{0}C**

4.Applying Boundary conditions

- Selection of injection location that is select the centre point on the face of sprue through here material will start to flow
- Clamping Force direction –Select Ydirection as our Mould closes in Y-Direction

Final step is to run the analysis and running the above parameters such as Flow results,Warp results,Cool results and pack results

5. As there are Four cases in our analysis above process parameters will be same for all cases but feed system design will be changing in all cases and analyzing the results of four cases

VII. Feed system details

Case 1:Existing feed sytem design with single trapezoidal runner and single pin point gate

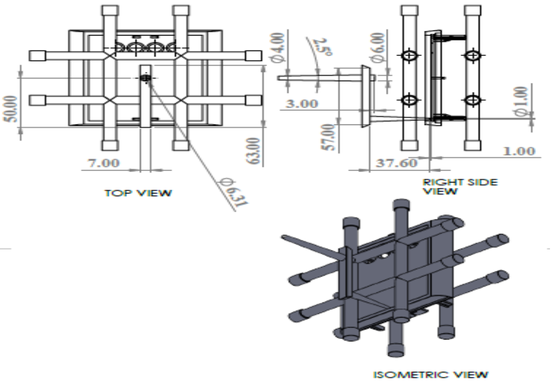

Fig. 4.1 a) Existing Feed system Design

Case 2:Modified double trapezoidal runner with two gates ,Dimesions of feed sytem remains same

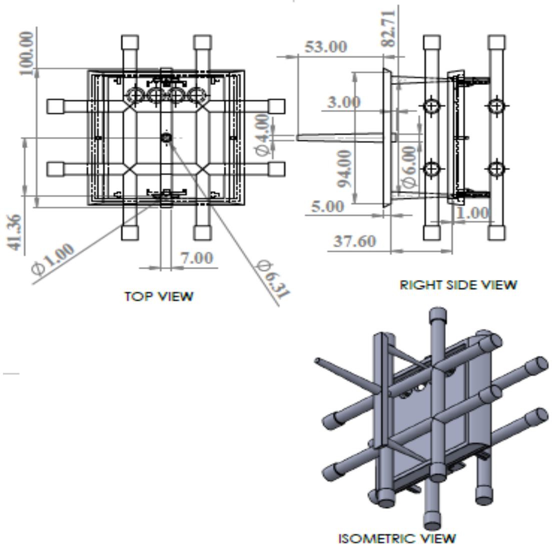

Fig. 4.1 b) Modified double trapezoidal runner with two gates

Case 3:Modified double trapezoidal runner with two gates but distance of gate is reduced from sprue

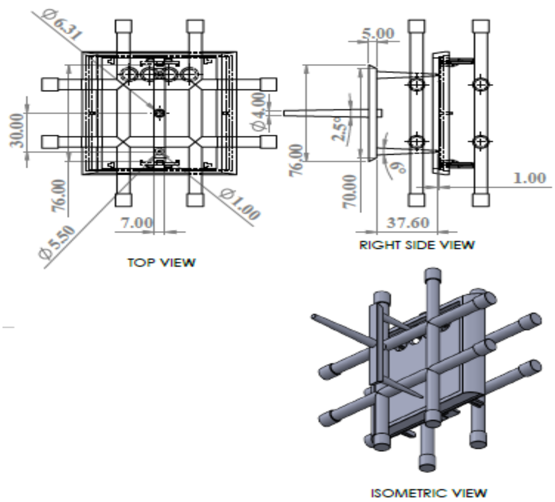

Fig. 4.1 c) Modified Runner and gate with reduced gate distnace

Case 4:Modified double Semi circular runner with two gates

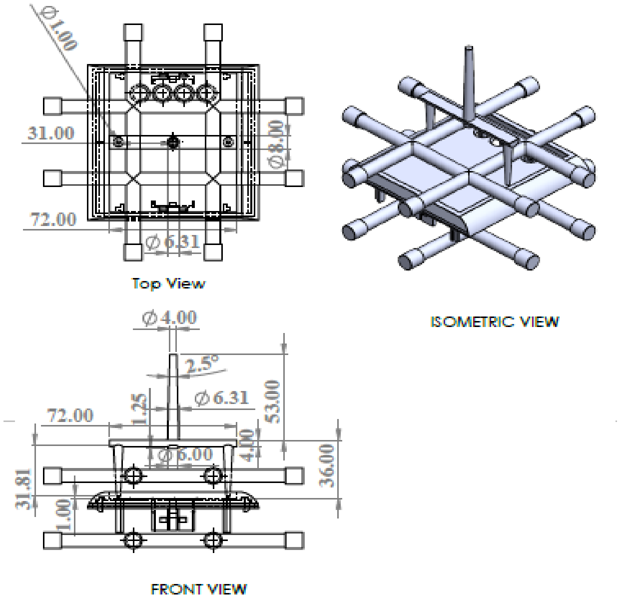

Figure 4.1 d) Modified Runner design

Above table shows feed sytem dimensions for all four cases
Discuss

	Case 1	**Case 2**	**Case 3**	**Case 4**
Sprue type	Cold, Circular	Cold, Circular	Cold, Circular	Cold, Circular
Sprue Dimensions	Start Diameter(4mm),End diameter (6.31mm) Tapered-2.5^0	Start Diameter(4mm), End diameter (6.31mm) Tapered-2.5^0	Start Diameter(4mm) ,End diameter (6.31mm) Tapered-2.5^0	Start Diameter(4mm),End diameter (6.31mm) Tapered-2.5^0
Gate Type	Pin Point	Pin Point	Pin Point	Pin Point
Gate Dimesion (mm)	Diameter (1mm),Length (1mm)	Diameter (1mm),Length (1mm)	Diameter (1mm),Length (1mm)	Diameter (1mm),Length (1mm)
Gate distance from Sprue	41.36mm	41.36mm	30mm	31mm
Gates and Runner placed in Direction	Z-Direction	Z-Direction	Z-Direction	X-Direction
Runner	Single Trapezoidal	Double Trapezoidal	Double Trapezoidal	Double half round
C.S area of Runner	$30mm^2$	$30mm^2$	$30mm^2$	$26mm^2$

VIII. Mould Flow Analysis Result

8. A Filling Time

Filling time is the time taken by the material to fill the Core and Cavity Impression .It indicates How much time material will take to fill the part in one shot of injection ,It also shows how plastic material will flow inside the cavity .At start of injection result is dark blue and place which fills last shows in red Colour.filling time depends on geometry,shape and size of material ,If geometry of material is large it will take more time to fill the part

Fill Time analysis Results for different cases shown in figures

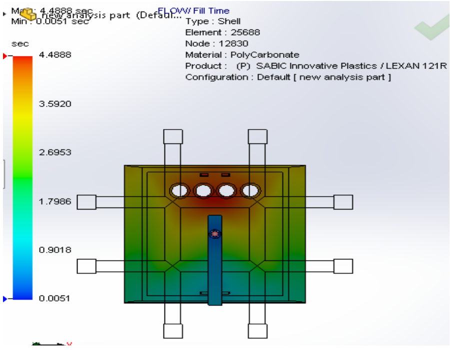

Fig .A case 1

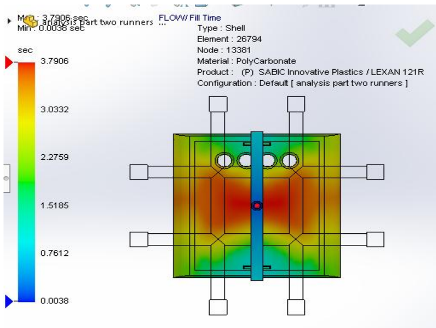

Fig. B case 2

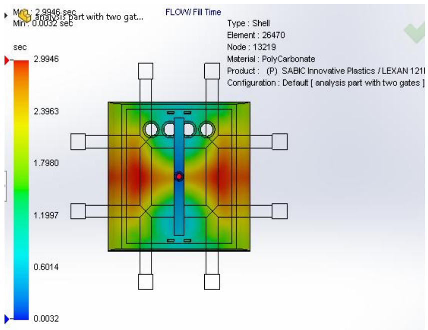

Fig. C case 3

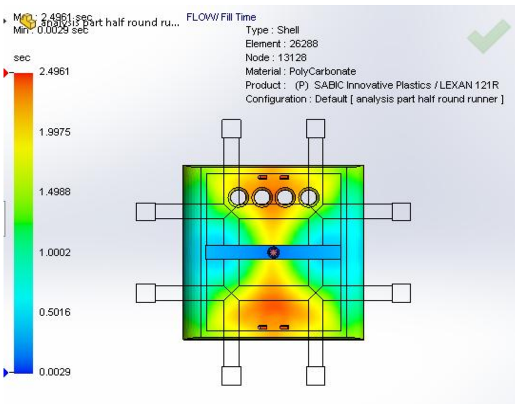

Fig. D case 4

8. B Cooling time

Cooling time is the time required by the part to cool and solidify to a temperature so that part can be ejected from the mould ,with the help of Coolant the Cooling time can be reduced also Cooling time depends on density,wall thickness of part.

Cooling Time analysis Results for different cases shown in figures

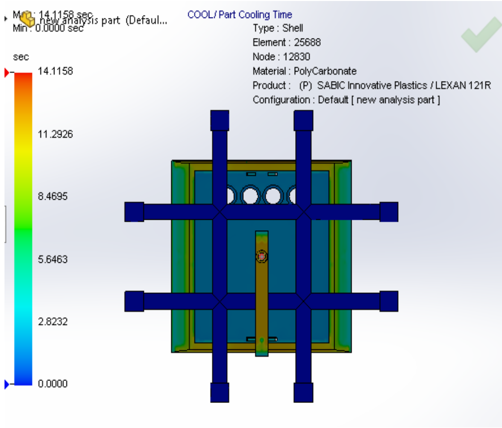

Fig. A case 1

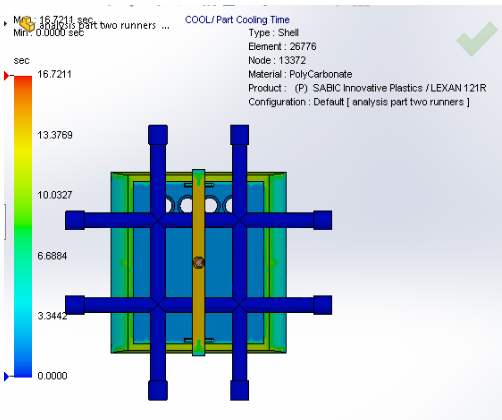

Fig. B case 2

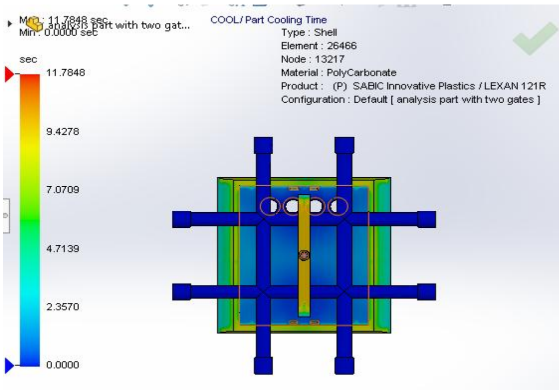

Fig. C case 3

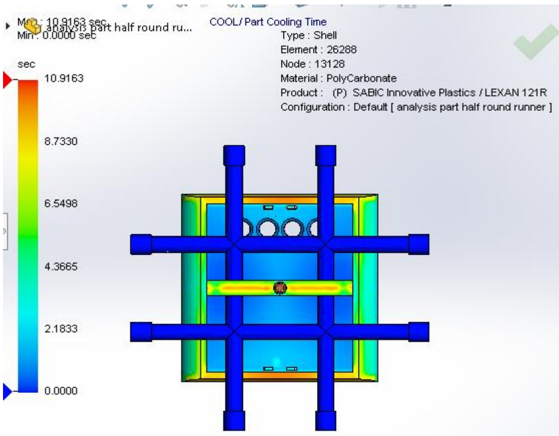

Fig. D case 4

8. C Pressure at the end of Fill

Pressure at the end of fill shows the pressure distribution in the cavity at the instant when the cavity is completely filled with Polymer

Pressure at the end of fill analysis Results for different cases shown in figures

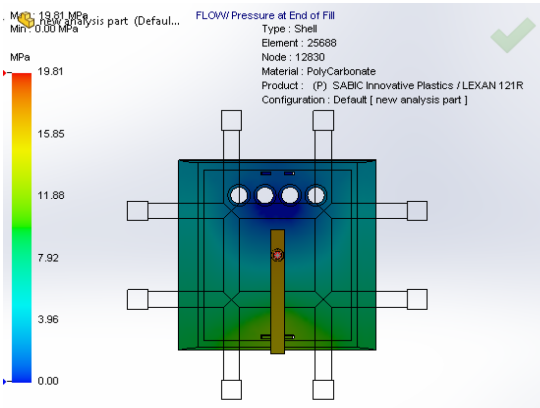

Fig. A case 1

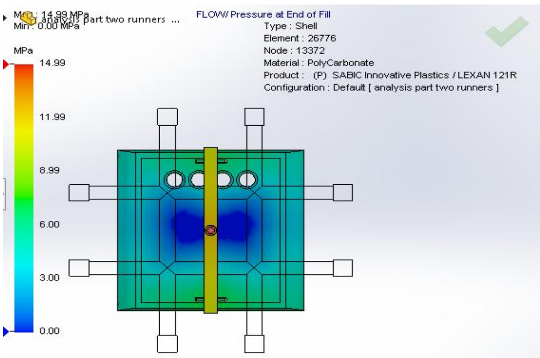

Fig. B case 2

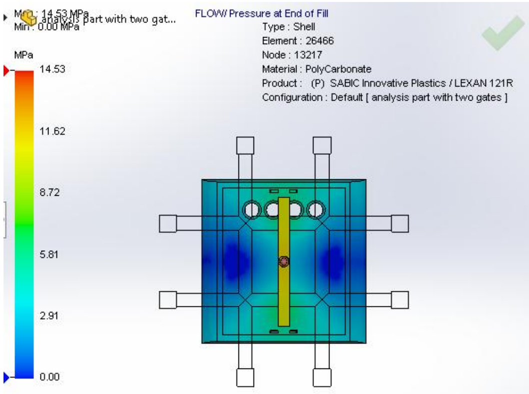

Fig. C case 3

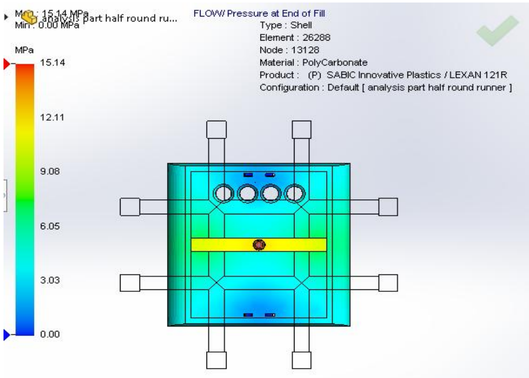

Fig. D case 4

8. D Volumetric shrinkage of part

Volumetric Shrinkage is caused by thermal contraction,which affects all polymers.It describes the extent to which the material changes in volume as it changes from a liquid to a solid.In general,plastics can shrink upto 25% during injection moulding process.Volumetric shrinkage contracts the part in all dimensions

Volumetric shrinkage for different cases shown in figures

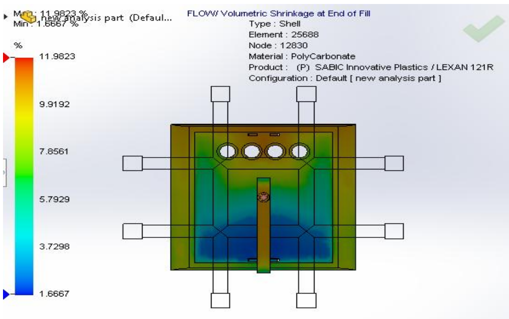

Fig. A case 1

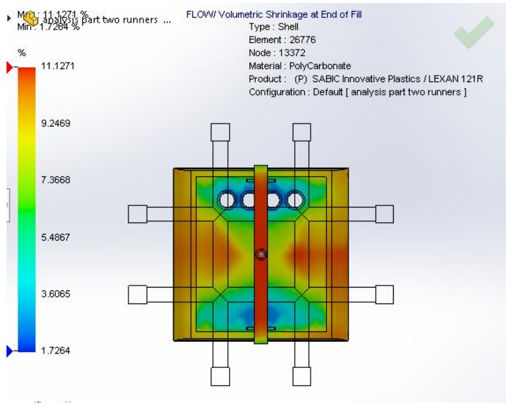

Fig. B case 2

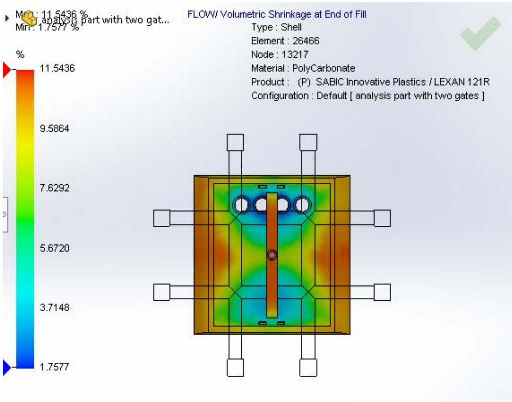

Fig. C case 3

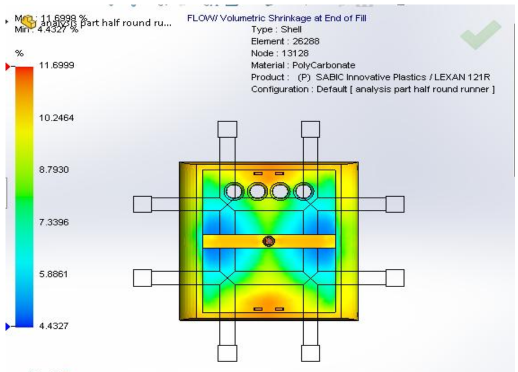

Fig. D case 4

IX. DEFECTS

9. A Weld lines

Weld lines occurs when two flow fronts meet each other,Weld line results displays the angle of convergence as two flow fronts meet.The presence of weld lines may indicate a structural weakness and /or a surface blemish

Weld lines for different cases shown in figure

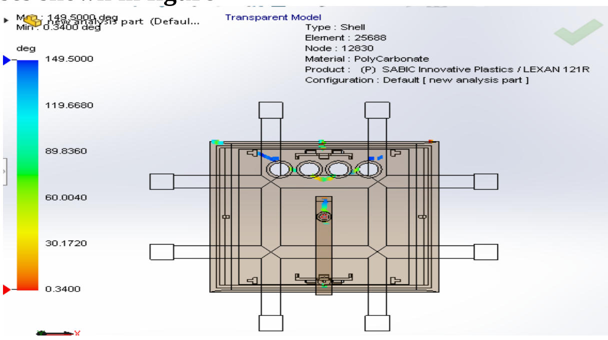

Fig. A case 1

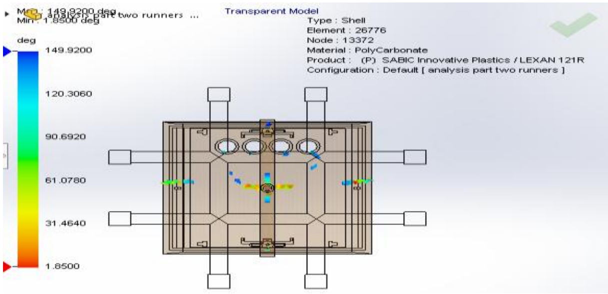

Fig. B case 2

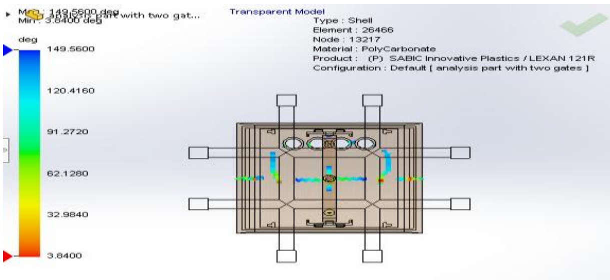

Fig. C case 3

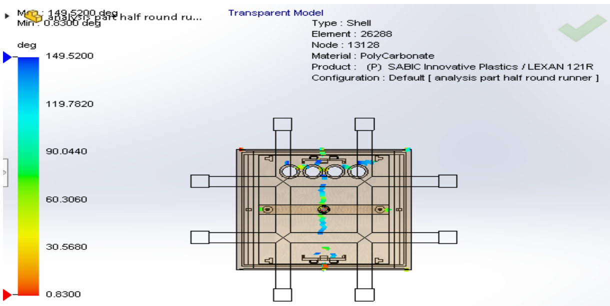

Fig. D case 4

9.B Air traps

An air trap occurs where the melt traps and compresses a bubble of air or gas between two or more converging flow fronts, or between the flow front and the cavity wall.

Air traps for different cases shown in figure

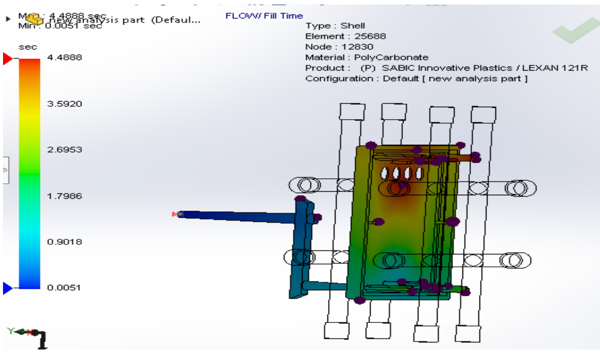

Fig. A case 1

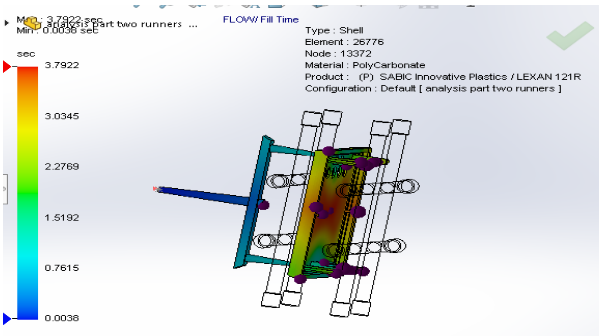

Fig. B case 2

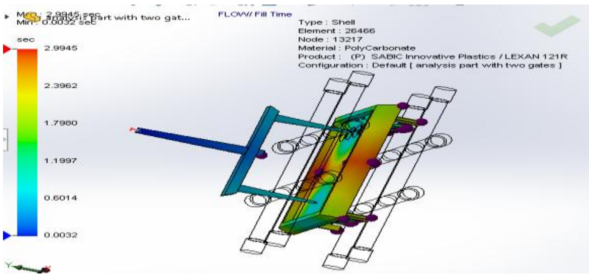

Fig. C case 3

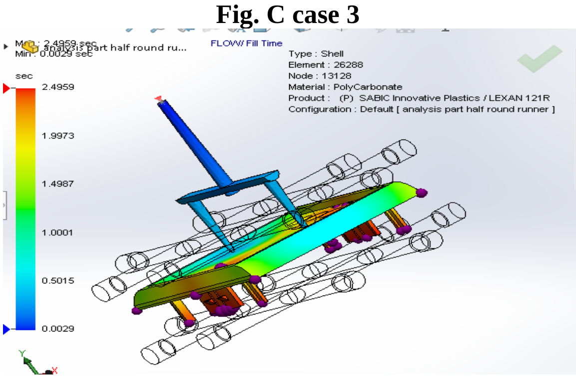

Fig. D case 4

9. C Sink marks

Sink marks are the area in moulded part where the surface is deformed into depression due to un even cooling of the material.They also caused by excessive temperature at the gate ,inadequate cavity pressure.

Sink marks for different cases shown in figure.

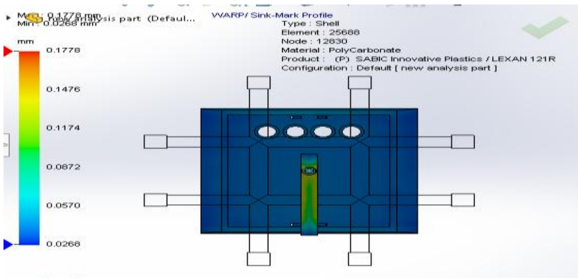

Fig. A case 1

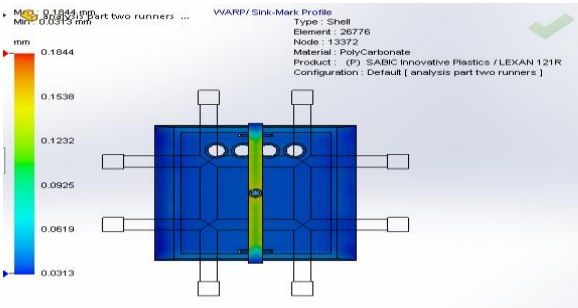

Fig. B case 2

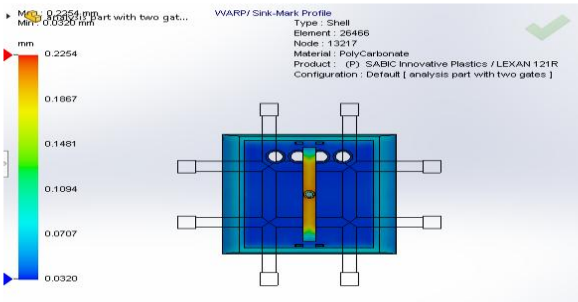

Fig. C case 3

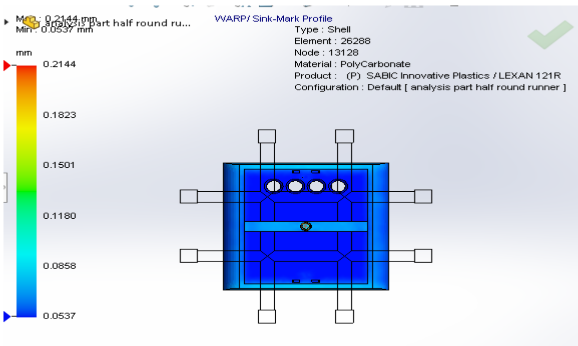

Fig. D case 4

X. RESULTS AND DISCUSSION

Description	Case 1-Feed system	Case 2-Feed sytem
Feed system weight (grm)	4.42	6.45
Filling Time(Sec)	4.49	3.79
Cooling Time(sec)	14.16	16.72
Pressure holding Time (sec)	5	4

Mold open Time (sec)	5	3
Ejection Time(sec)	3	3
Cycle Time (sec)	31.65	30.51
Calculated injection pressure (MPa)	57.270	57.270
Required Injection pressure(MPa) or Pressure at the end of fill	19.81	14.99
Volumetric shrinkage Range	1.66% to 11.98%	1.72% to 11.12%
Weld lines	Less	More
Air traps	More	Less
Sink marks depth Range(mm)	Min-0.027 Max-.177	Min-0.313 Max-0.18
Clamp Force in all directions(Tonns)	X-2.68 Y-5.51 Z-2.41	X-2.2 Y-3.75 Z-2.3

Description	**Case 3-Feed sytem**	**Case 4-Feed system**
Feed system weight (grm)	5.45	4.43
Filling Time(Sec)	2.99	2.45
Cooling Time(sec)	11.78	10.91
Pressure holding Time (sec)	2.5	2.5
Mold open Time (sec)	3	3.5
Ejection Time(sec)	3	3
Cycle Time (sec)	23.27	22.36
Calculated injection pressure (MPa)	57.270	57.270
Required Injection pressure(MPa) or Pressure at the end of fill	14.53	15.14
Volumetric shrinkage Range	1.75% to 11.54%	4.43% to 11.7%
Weld lines	Less	More
Air traps	Less	More
Sink marks depth Range(mm)	Max- 0.225 Min-0.032	Max-0.214 Min-0.053
Clamp Force in all directions(Tonns)	X-1.50 Y-2.78 Z-1.76	X-1.75 Y-4.51 Z-1.52

The Comparison of existing feed system that is case 1,along with newly design feed systems case2,case3,case4 on various parameters

1. Feed system weight for case 1 is 4.42 ,feed sytem weight forCase 2,case3,case4 are 6.45,5.45,4.43 gm .Among these existing feed system design has less weight and newly designed case 2 feed system has more weight,In newly designed case 4 feed system weight is 4.43 gm which is also very less and hence less wastage of material.
2. Filling time for existing feed system is 4.49 sec and newly designed feed sytem case2,case3, case4, are 3.79,2.99,2.45 secSingle trapezoidal runner and single gate fills the material within more time hence in our design for three cases material fills faster Among all design case 4feed system fills material faster.
3. Cooling time for existing feed system is 4.16 sec and for newly designed feed sytem case2, case3,case4, are 16.72,11.78,10.91 sec Among these cases case feed system has less cooling time for the part
4. Cycle time is the combination of Filling time,Cooling time,Pressure holding time,Mold open time,Ejection time ,Cycle time for existing feed sytem is 31.65 and cycle time for new feed system case2,case3,case4 are 30.51,23.27,22.36 sec.Cycle time plays an important role in injection molding if cycle time of part is less than production time of part is increased.case 4 feed system has less cycle time
5. Clamping Force along y direction in which mold closes for existing feed system 5.51 tonn and for newly designed feed system cases are 3.75,2.78,1.52 tonns
6. Calculated Injection molding pressure for all cases are same .This injection pressure 57.2708 is obtained by using Windsor Machine data standard table

XI. CONCLUSION

- This paper concludes that by designing new feed system that is case 2,case3,case4 and comparing it with existing feed system case1.Then by doing Mould flow analysis we can select optimum feed system design which has less cooling time,filling time,cycle time,surface defects such as volumetric shrinkage,air traps,weld lines
- Before actual manufacturing of tool with the help of mould flow analysis software such as solidworks we can analyze the flow of material and its characteristics and selecting best optimized feed system with less wastage of material and defects for actual manufacturing of plastic part
- Now among these four cases of feed sytem design case 3 and case4 are more optimized than the case 1 and case2 feed system
- Finally selecting best feed system among case 3 and case4
- In case 4 feed system cycle time is less 22.36 sec that is0.91 seconds less than case3 if we have to increase production rate than case3 feed sytem is best
- Cooling plays an Important role in injection moldingIt comprises of 50 to 75% of cycle time in Injection molding if we can reduce cooling time than we can increase production time of the part
- Cooling must be done properly uneven cooling leads to surface defects such as warpage, sink marks
- More cooling leads to increase in cycle time of part hence cooling must be done optimum
- In case 4 feed system Cooling time is less 10.91 sec that is 0.87 seconds less than case 3 feed system
- Case 4 feed system has less cycle time but surface defects are more
- But if we want good quality of part with less surface defects case 3 feed system is best
- Case 3 feed sytem has less air traps and less weld lines on the surface of part

REFERENCE

[1]. Solidworks Mould flow insight material data ware house

[2]. Manmit salunke and rushikesh kate "Injection molding methods design,optimization,simulation of plastic toy building block by mold flow analysis" International journal for Mechanical engineering and technology (IJMET) ,Volume 6,issue 6 ,June (2015),pp.33-42 ,ISSN 0976-6340 (print),ISSN 0976-6359 (online)

[3]. Dr.Mp singh and Akhilesh jaguri ,"Gating sytem design optimization for injection moulding" International journal of Mechanical and industrial technology (IJMIT),Volume 3, issue 1,pp☹156-163),ISSN 2348-7593 (online)

[4]. R Hussin,R Mohd saad,Rhussin,M.R.M Hafiezal,M.a.Fairuz, "Optimization of the plastic injection molding parameters for sport equipment by using design of experiment",International review of Mechanical Engineering (IREME),Volume 7,N.3,ISSN 1970-8734

[5]. Z Rutkauskas ,A.Bargelis , "Knowledge-based method for gate and cold runner definition in injection mold design" ISSN 1392-1207.MECHANIKA.2007.Nr4(66)

[6]. Mubarak Abdullah Ali Mohsen Alhadad,Quan Wang , "CAE analysis and stsructure optimization Design of injection mold for charge upper cover", International Journal of engineering research and technology (IJERT),Volume 8,Issue 11 November 2019,ISSN:2278-0181

[7]. K.R .Yatish kumar and P,Nagaraja.R, "Significance of mold filling analysis for finding optimal gate location in injection molding process for Bobbin" International journal of scientific and research publications,vol.4,no.4,pp. 1-8,2014

[8]. Injection mold design handbook by Rgw pye Fouth edition

[9]. GTTC mold design handbook

12.AUTHOR'S INFORMATION

STUDENT

Iam **MOHAMMED ABDUL JAWAD** doing Mtech(Tool Engineering) In Government Tool Room and Training centre,Bengaluru.Currently Done final semester project work on Optimization of feed sytem design under the guidance of Dr.R NAGARAJA Ph.D., Mtech in Tool Engineering

PRINCIPAL/INTERNAL GUIDE

Dr. NAGARAJA R obtained his Bachelor's degree in Mechanical Engineering from Mysore University, Chickmagalur in 1988 His M.tech and Phd in tool Engineering was obtained from Visvevaraya Technological University, BelgaumHe started his career at Govt. Tool Room &Training Centre in the year 1993 and he has 31 years of experience in various departments such as Tool Design, Production and Training. He is presently working as Principal for PG Studies in the same organization. His areas of interest are too design, development, validation and natural convection heat transfer in enclosures.