Design for Manufacturing - 17ME744

IAT 3

Answer all the 3 questions (50Marks)

18/12/2020 1.30 pm to 3.00pm

1. Explain the design recommendations for the following forging variables.

   (i) Parting line (ii) Draft (iii) Radii (iv) Rib (v) Recesses *(16 marks)*

2. Explain the design recommendations for powder metallurgy with suitable sketches. *(16 marks)*

3. (i) Explain the injection molding process with a schematic set-up. *(8 marks)*
   (ii)Explain the design recommendations for injection molding with suitable sketches. *(8 marks)*
   (iii) Add a note on injection molding materials. *(2 marks)*

Q1. (1) Parting line: i) As the die halves come together and confine metal in their cavities, their mating surfaces define a parting line around the edges of the forging.

ii) Parting line is indicated on the drawing and determining its location it a critical step in forging design.

iii) Ideally parting line will lie in one plane perpendicular to the axis of die motion. Sometimes it can be located so that one die half will be completely flat and it will surround the largest projected area of the piece. If the parting line cannot lie in one plane, it is desirable to preserve symmetry so as to prevent high thrust forces on the dies and the press.

(2) Draft: (1) Die impressions are tapered so that forging can be removed from their dies.

(2) So the forged surfaces that lie parallel to die motion are tapered.

(3) This taper is called draft. This draft also promotes flow into relatively deep die cavities. Standard draft angles will be specified on for all affected surfaces on a forging, which simplifies tooling for die sinking.

(4) It is also conventional to call for matching draft on both die halves to make surfaces of internal depth meet at parting line.

(3) Radii: (1) Forging are designed with radii on all their internal corners except at the parting line. It would require a sharp internal angle in the die to form a sharp corner on the forging.

(2) This is stress raiser and excessive pressure is required to fill sharp corners.

(3) Common practice is to provide full radii at the each edges of all ribs and same radius on each corner of a boss, web, or other shape.

(4) Rib: Forging becomes difficult when large amounts of metal must be moved out of a relatively thin web into deep ribs and high bosses. It is helpful to taper such webs towards the ribs and bosses.

(5) Recesses: Deep recesses are easier to forge if they have spherical bottoms.

Q2: (1) The design must be such that the part can be ejected from the mould or die. Parts with straight wall are preferred. No. draft should be required for the ejection of a part from a lubricated die.

(2) In designing the part, consideration should be given to the need for the powder particles to flow properly into all parts of the mould or die. Therefore, thin walls, narrow splines, or sharp corner should be avoided (should be thicker than 0.762 mm).

(3) The shape of the part should permit the construction of strong tooling. Dies and punches should have no sharp edges. Reasonable clearance must be provided between the top and the bottom dies during pressing.

(4) Since pressure is not transmitted uniformly through a deep bed of powder, the length of the part should not exceed about 2 and half times of the diameter.

(5) Very close tolerance in the direction of compression should be avoided.

(6) Shape of the parts should be kept as simple as possible and should contain with few levels and axial variations. Holes should not be designed in the direction of pressing.

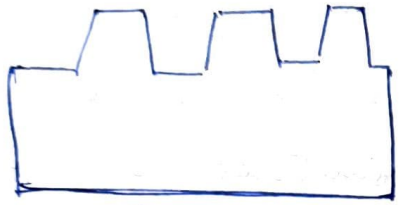
(7) Provide sufficient wide dimentional tolerance whenever possible. Wide tolerance means that the part can be made more economically with a longer tool life.

(8) In designing flat section of high density, enough section thickness should be provided otherwise the punch may break under pressure.

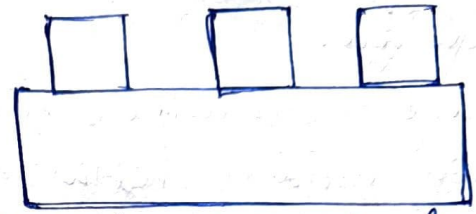
(9) Parts made through powder metallurgy may be bonded by assembling in the green condition and then sintering together to form a bond assembly.

(10) As far as possible, abrupt changes in the section thickness should be avoided

(a)

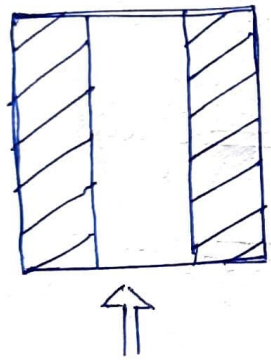


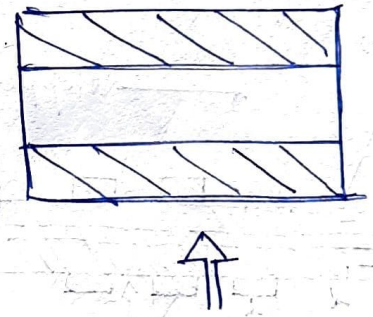
Draft is required when no lubricant is used.



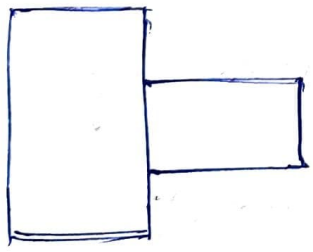
Draft is not required when lubricant is used.

(b)

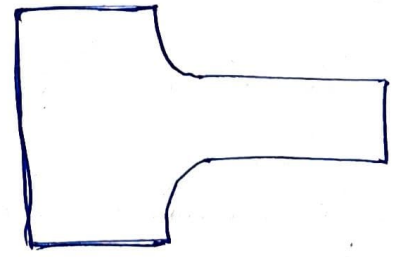




(c)



Not this



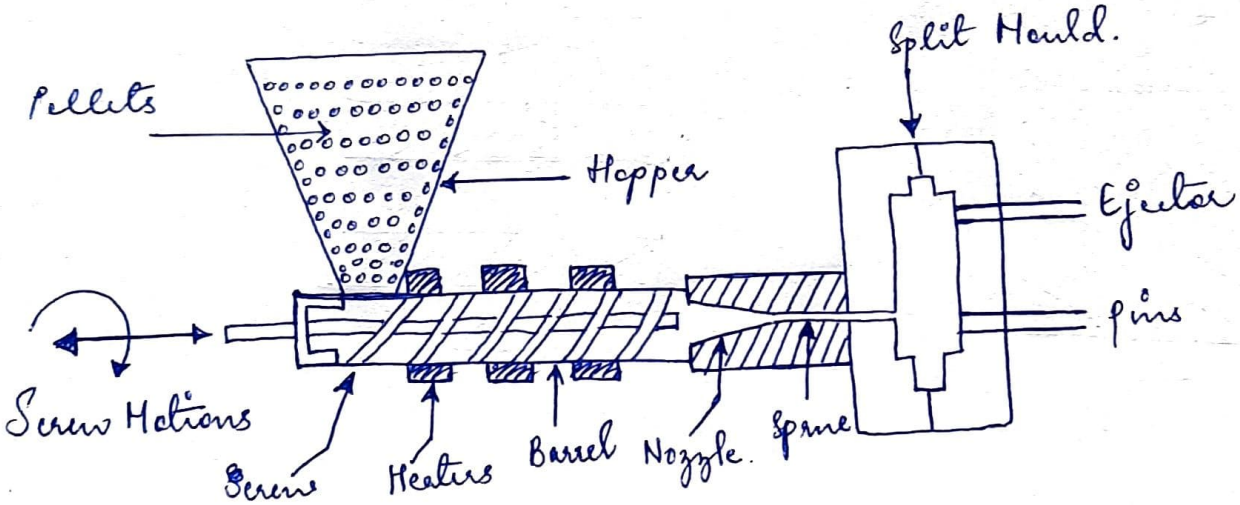
This.

Q3.

1.> ① Design for gating and feed system for the die is crucial to ensure complete die fill. It is important to design the moulding so that solidification does not prevent complete mode filling.

② Mold must be designed in such a way that the solid part can be ejected without distortion. By considering proper orientation at the beginning, it may be possible to avoid expensive mold cost. If possible, design the part so that it can be ejected in the direction of mold closure.

③ To minimize the shrinkage filler like glass fiber, wood flour, are. added during molding, with same part geometries, post mold shrinkag can lead to generation of high residual stress.



Q3. ii> ① Streamlined flow will avoid gas pockets in heavy-sectioned areas.

② Provide draft, which is required for easy ejection of molded parts from the mold cavity. A taper of 1 to 4 degree is usually used for polymers, but tapers of less than 1 degree can be used.

③ Uniform thickness: Design the mains wall of uniform thickness with adequate tapers or draft for easy release from the mold. This will minimize part distortion by facilitating even cooling throughout the part.

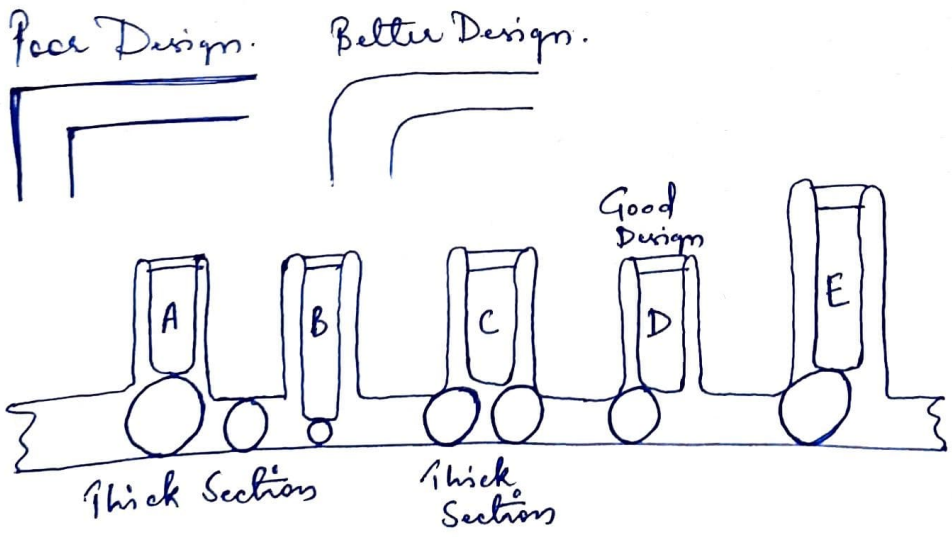
(4) Choose the material and main wall thickness for minimum cost. Note that a more expensive material with greater strength or stiffness may often be the best choice. The thinner wall this choice allows will reduce material volume to offset the material cost increase. More important, the thinner wall will significantly reduce cycle time and hence processing cost.

(5) If possible, align projections in the direction of molding and at right angles to the molding direction lying on the parting plane. This will eliminate the need for moldi mechanisms.

(6) Avoid depressions on the inner surfaces of the part, which would require moving core pins to be built inside the main core.

(7) If possible, design external screw threads so that they lie in the molding plane. Alternatively, use a rounded or rolled type thread-profile.

(8) Presence of holes disturbs the flow of the material during molding and weld line occurs in the side of the hole away from the direction of flow.



Q3. Ans: Materials for is fed into a heated barrel, mixed (using helical shaped screw), and injected into the mould cavity where it cools and hardens to the configuration of the cavity.