COSMETIC FINISH STANDARD
PLASTIC INJECTION MOULDINGS
DSF 3.2.10

Definition:

Injection moulding is a manufacturing process using plastic which has passed through a heated barrel (moulding press) melted, then forced under pressure into a mould cavity (tool) where it cools & hardens to the shape of the cavity. The moulding is then ejected from the tool.

ASM (DEK) Application:

This standard applies to any ASM (DEK) designed component used on a machine, or suppllied as a spare by the vendor.
All components must meet the criteria set out in this standard, unless a concession has been granted by ASM (DEK).
Where conflict is found between this standard and any other specification, the priority should be as follows:-

1st Customer Specified Requirement.
2nd ASM (DEK) Engineering specification.
3rd Purchasing contract
4th This standard

This standard is not an exhaustive list, any component displaying a non-specified sympton which may cause a customer issue should be reviewed only by authorised ASM(DEK) personnel.

Only upon this approval can the component be released from ASM (DEK) or its vendor.

Requirements for Plastic Moulded Parts:

Once approved, moulded parts shall be consistent in appearance, size & material. The moulder shall not change the material or process without prior approval from ASM/DEK Engineering.

- **Flash Control:** (see image 1). Flash & mismatch at parting lines to be 0,3mm max. Holes and apertures containing flash to be within tolerance.
- **Ejector Pins and Gates:** (see images 1& 2). Ejector pin marks shall be flush to 0,3mm max. sub flush (minus) at component profile. Gates shall be trimmed flush to 0,4mm above (plus) or below (minus) at component profile. Except for structural foam moulded components, gates shall be trimmed flush to 0,4mm above (plus) component profile.

COSMETIC FINISH STANDARD
PLASTIC INJECTION MOULDINGS
DSF 3.2.10

Requirements for Plastic Moulded Parts: (cont'd)

- **Ejector Pins and Gates (cont'd):**

Placement of ejector pins, parting line(s) & gate(s) to be agreed with both toolmaker and ASM/DEK Engineering, prior to machining.

- **Cavity Ident. :** (see image 1)

For multiple cavity tools, each cavity is to be identified by a unique number. For example, on a four (4) cavity tool, each individual cavity would be identified by using only one number from the following: 1, 2, 3 & 4.
It is common practise to place the cavity number adjacent to any Part Number. This will be agreed with both toolmaker and ASM/DEK Engineering.

- **Surface Quality:**

Blemishes such as weld & flow lines or surface imperfections including sinkage, short shot and porosity will need to be agreed by ASM/DEK Engineering prior to dispatch.

- **Colour:**

The colour of a component is determined by the material formulation and will not normally fluctuate within a batch. Where a difference is noted then the following procedure should be followed to determine acceptability:-
Mount the item to be inspected onto the machine. If not possible, ensure that it is in the same orientation as used on the machine.
Set the distance between the item and the inspector to 2,5metres.

1. Ensure the lighting system in the inspection area is normal and that light intensity is in the region of 800 LUX.
2. Without using any optical magnifying equipment, look at the item with the naked eye only. (The inspector is allowed to wear their normal prescription glasses but these must be clear transparent glasses and not coded glasses or sun glasses).
3. If the discoloration is visible within 30 seconds, and the item is set to the correct distance; it can be considered rejected.
4. If the discolouration is hardly visible or completely invisible within 30 seconds and the item is set to the correct distance; it can be considered acceptable.

COSMETIC FINISH STANDARD

PLASTIC INJECTION MOULDINGS

DSF 3.2.10

Glossary:

Gate: A small aperture where molten plastic enters to fill the mould cavity.
Ejector Pin: Used in the mould tool to remove the moulding.
Flash: Excess material that extends from the moulding.
Weld lines: A visible line identifying where two flow fronts (cold) meet but do not merge (knit). These lines usually occur around holes or obstructions on the opposite side to the flow and cause local weak areas in the moulded part.
Flow lines: Visible semi-circle parallel knit lines rippling out from the gate causing a local weak area in the moulded part.

IMAGE 1

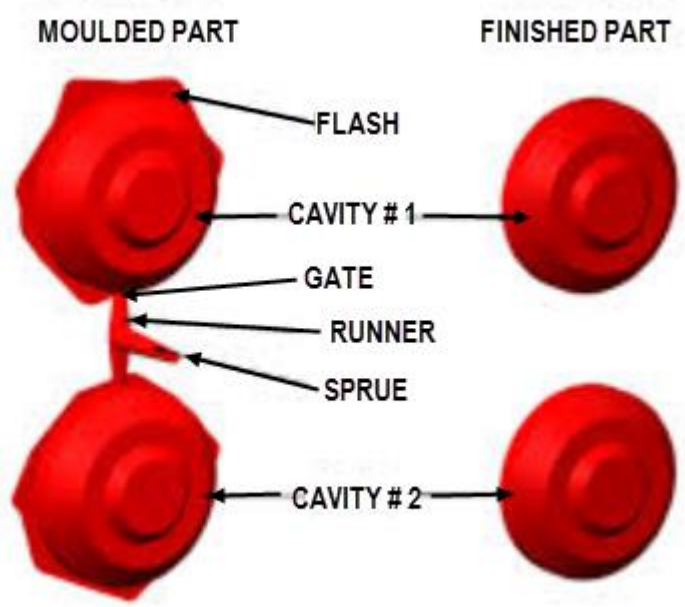
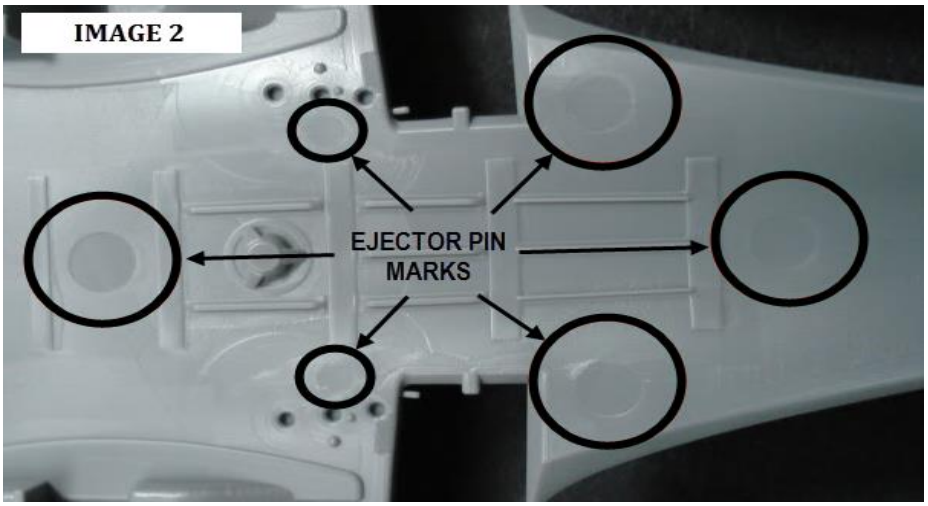


IMAGE 2



Issue	01	02	03	04	05	06	07	08	09
DCR	41730	44734							