

THERMOLAST K in Injection Moulding: Processing

1. Processing Parameters

The following sections provide information and recommendations for processing THERMOLAST K. Please keep in mind that your experience with your machines and the particular material processed should always be taken into consideration as well. As THERMOLAST K compounds are frequently custom-developed to individual customer requirements, differences in processing can result.

Our applications engineers will be happy to provide any advice you may require and to come to your plant upon request to observe processing of sample materials.

2. Barrel and Melt Temperatures

When processing standard THERMOLAST K compounds, adjust the barrel temperature such that the melt temperature is between 180° C (355° F) and 220° C (425° F) (max. 250° C (480° F)). To do this, increase barrel temperature progressively in all zones in steps of 10° C (50° F) to 20° C (65° F) starting from the hopper side until the melt temperature is in the desired range.

The temperature at the shut-off nozzle should be the same or 10° C (50° F) below the barrel temperature of the last heating zone. Overheating, i.e. thermal decomposition of the material, is easily recognized as indicated by a distinctive odour.

Exact temperature recommendations for THERMOLAST K standard compounds are given in the processing instructions in the appendix. In the case of co-injection moulding, special procedures are required as is the case with other materials.

3. Mould Temperature

The mould temperature should normally be between 25° C (75° F) and 40° C (105° F). If the parts produced are thin-walled, we recommend mould temperatures between 40° C (105° F) and 60° C (140° F).

More information on mould temperatures, e.g. for co-injection moulding with 2 or more components, can be found in the appendix. Our applications engineering department will also be pleased to provide you further information upon request.

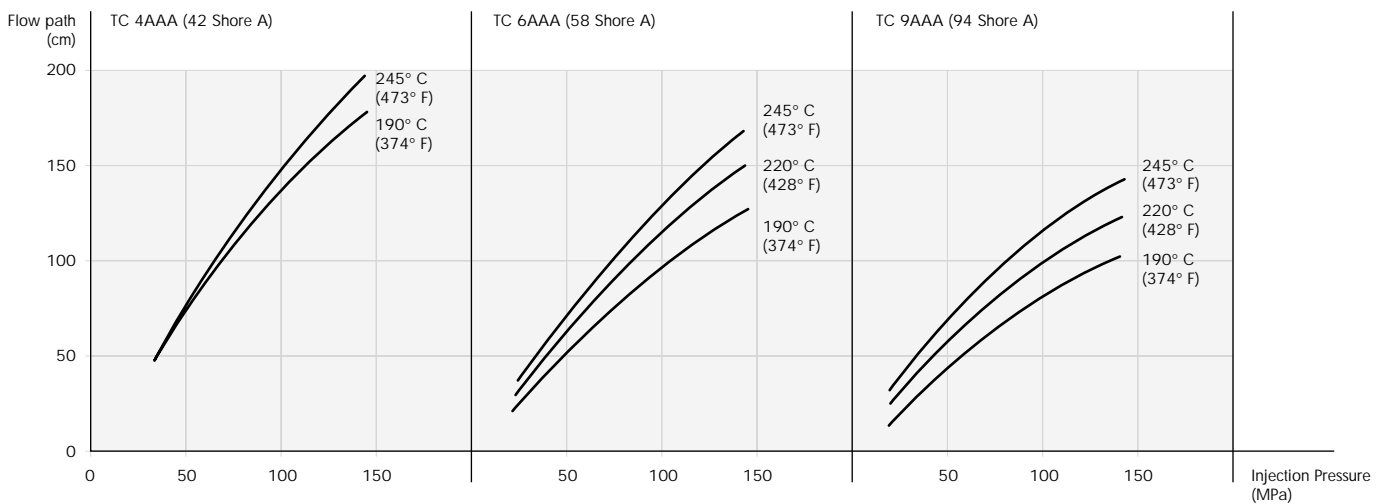
THERMOLAST K in Injection Moulding: Processing

4. Injection Pressure and Injection Rate

The cavity should normally be filled under high injection pressure and at a high injection rate to facilitate long flow paths. This applies especially if the part moulded has very low wall thickness (rule of thumb: injection rate should be in the range of approx. 100 – 200 g/s (0.22 – 0.44 lbs)). With these parameters the optimum mould filling is set by adjusting the metering.

The dependence of flow behaviour on temperature and injection pressure is illustrated below in flow path diagrams generated by testing three THERMOLAST K standard compounds.

The diagrams show that the flow properties of THERMOLAST K compounds are far more dependent on injection pressure than on temperature.



- Injection process conditions:
- ▶ Injection time 10 seconds
 - ▶ Spiral flow path in 6 mm (0.236 in.) D semicircular channel
 - ▶ Mould temperature 45° C (113° F)

THERMOLAST K in Injection Moulding: Processing

5. Back Pressure and Screw Speed

Usually we recommend a back pressure of approximately 20 – 50 bar (285 – 710 psi). The screw speed should be between 25 and 75 rpm.

Effects of higher back pressure:

- ▶ improves distribution e.g. of colour batches and other additives
- ▶ increases melt homogeneity
- ▶ plasticizing time is increased for a given screw speed
- ▶ reducing the screw speed also has a positive effect on homogeneity

Effects of lower back pressure:

- ▶ creates insufficient shear energy and inhomogeneous melt
- ▶ as a result, higher-melting components in the compound may not be plasticized.

6. Hold Pressure and Hold Time

The mould should be filled initially without applying hold pressure. Low hold pressure can then be applied to prevent sink marks and to guarantee 100% filling. The hold time should be set as short as possible.

Possible effects of high hold pressure:

- ▶ overfilling of cavities
- ▶ difficulties in demoulding
- ▶ de-lamination
- ▶ deformation around the sprue
- ▶ improved adhesion at flow lines
- ▶ additional information concerning coinjection moulding is provided in "THERMOLAST K in Injection Moulding/Special Information on Coinjection Moulding".

Possible effects of low hold pressure:

- ▶ sink marks in thick-walled parts
- ▶ increased shrinkage