Arnitel® TPE-E

Recommendations for Injection Molding

Machinery

Standard screw plunger type injection molding machines must be used. The machine size and cylinder diameter should be such that the product weight is within a range of approximately 40 to 70% of the maximum shot capacity.

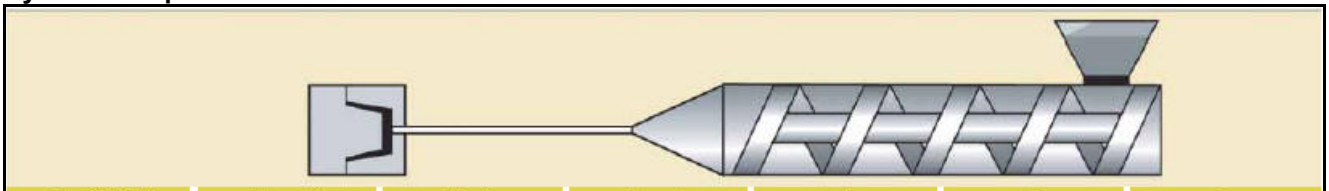
Screw: Standard three-zone screws with a L/D ratio from 17 to 23 and a thread depth ratio of about 1:2 yield excellent results. A non return valve at the end of the screw is recommended.
Screws for processing PVC must not be used for Arnitel®.

Nozzle: It is recommended to use a short nozzle and a wide bore (3 mm or more) to minimize frictional heating and pressure losses.

Hopper: The hopper should be equipped with a tightly-closing lid which should be kept closed during processing to keep the granules dry and free from dust.

Processing Conditions

Cylinder temperatures:



Arnitel® E	Mould	Melt	Nozzle	3	2	1
	°C	°C	°C	°C	°C	°C
EM400	20 - 35	220	215	210	205	200
EM460	20 - 35	220	215	210	205	200
EL550	20 - 50	235	230	225	220	220
EL630	20 - 50	240	235	230	225	220
EL740	20 - 50	245	240	235	230	225
Arnitel® P						
PL380	20 - 35	230	230	225	220	215
PL460	20 - 40	235	235	230	225	215
PL471	20 - 50	240	235	235	235	225
PL581	20 - 50	240	240	230	225	215

Polymer temperatures of 230°C (Arnitel® EM400 and EM460) and 260°C (EL550, EL630 and EL740) are the maximum values for longer residence times in the cylinder. Residence times must be kept as low as possible.

Mold Temperature: For thin walled products a mold temperature of 50°C is recommended. For thick walled products, 20°C can be maintained. Higher mold temperature facilitates flow at the cost of cooling time.

Injection Pressure: Must be high enough for mold filling. The trend of flow length with hardness for Arnitel® EL and Arnitel® PL grades follows an inverse relation for the same injection pressure, whereas grades for Arnitel® EM follow a direct relationship.

Injection rate: Injection rate must be high for filling the mold quickly to avoid premature freezing during molding. For certain conditions, a moderate injection rate may be required.

Holding pressure and time for follow-up pressure: Holding pressure must be 40 to 70% lower than injection pressure to ensure that volumetric shrinkage of the cooling melt can be compensated. The holding pressure must be set high enough to prevent sink marks.

The holding pressure should be sustained until the gate freezes. Sink marks or shrinkage voids indicate that the holding pressure time is too short. The holding pressure time should be prolonged proportionally as wall thicknesses and gate dimensions increase.

Excessively high holding pressures should be avoided since they may cause residual stresses in the product or visible burning.

Back Pressure and Screw Speed: It should be set just high enough to ensure that the melt is free from air bubbles, that the screw plasticizes evenly, and that the product weight is constant.

A hydraulic back pressure of approximately 3 to 6 bar is sufficient. A screw speed of 30 to 100rpm is generally suitable without generating excessive shear heating.

Metering: A small buffer of 2 to 5 mm is recommended, since a large buffer might lead to loss of pressure and to prolonged residence of the melt in the cylinder.

Mold Design

Gating System: All common gating systems may be used, including cone, pinpoint, tunnel, film, fan, and ring gates. Externally heated hot runner and semi-hot runner systems also qualify, but require efficient heating and very accurate temperature control to avoid freezing or overheating the material.

Gate Location: Gates must be located in such a manner so as to have uniform properties of the product molded. Preferably, gates must be located on the thickest wall section and in such a way that the product fills uniformly.

Gate and Runner Dimensions: The cross-section of the runners should preferably be circular. Where this is not feasible, the best compromise is a trapezoid. Recommended runner and gate dimensions for various wall thicknesses are given below. For products with a wall thickness exceeding 3-5 mm, a full sprue gate with a diameter of about three-quarters of the largest wall thickness is preferred. A short sprue cone with a taper of at least 1°30' is recommended.

Wall thickness *(mm)*	**Gate diameter/ length** *(mm)*
0.7 - 1.2	0.7 - 1.0 / 0.8 - 1.0
1.2 - 3.0 3.0 - 5.0	0.8 - 2.0 / 0.8 - 1.0 1.5 - 3.5 / 0.9 - 1.0
> 5.0*	3.5 - 6.0 / 0.8 - 1.0

*(*wall thickness >5mm must be avoided).*

Venting: Special attention should be given to effective mold venting. Venting is effected by vents (approximately 0.02 mm) in the mold faces, or via existing small channels such as those around ejector pins and cores. Vents should be located in the mold, at the end of the flow paths.

Ejection: Ejection of the part from the mold is facilitated by ejector pins, plates or rings. Ejection must not cause damage or deformation. In view of Arnitel® 's flexibility (particularly the softer types), that part of the product in contact with the ejector should be under uniform load. A fairly large ejector face is therefore required.

Cooling: The cooling system is an important part of the mold and needs to be configured with scrupulous care. The product must be cooled rapidly and uniformly to prevent warpage and long cycle times. Here are some recommendations for an effective cooling system design:

- plan a sufficient number of generously dimensioned cooling channels, configuring them symmetrically around the molding cavity and in the closest possible proximity to it. The distance between channels and to the mold cavity should be within 1 to 1.5 times the channel diameter

- avoid long cooling circuits. Compact, independent circuits are most effective, and parallel cooling is superior to stepped cooling

- incorporate one or more thermocouples or sensors in both mold halves to provide a check on mold temperature.

Material Handling

DSM Sales Offices