MEXMID A GF15 H NC

Description	Polyamide 66 medium viscosity with 15% glass fiber reinforced, heat stabilized		
Color	Natural Color	**Additional formulations**	
Processing	Injection	HR - Resistance to hydrolysis	EL - High impact
Norm	-	Viscosity from 2.4 to 3.3	IB - Hybrid Mineral+GF
Norm	-	UL94 - Flame	UV - Light stabilized

Applications: Lamp socket housings, cooling fans, insulating profile for aluminium window frames, water containers for automotive cooling systems, as well as electrically insulating parts.

Mechanical Properties	**Values**	**Unit**	**ISO**
Density	1.23	g/cm^3	1183
Filler Content	15	%	3451
Reletive viscosity (1% in 96% H_2SO_4)	2.7 ± 0.10	-	307
Melting Point (DSC)	262	° C	3146
Mechanical Properties	**Dry/Wet**	**Unit**	**ISO**
Tensile elongation at break	3/10	MPa	527-2
Tensile strength at break	130/85	MPa	178
Flexural Modulus	5200/4000	MPa	178
IZOD Impact strength, notched (23° C)	8/11	KJ/m^2	180 1eA
Thermal Properties	**Values**	**Unit**	**ISO**
HDT method A (1.820 MPa)	250	° C	75-1
Flammability	**Values**		
Flame rating at 3.2 mm	HB		UL94

Processing Conditions				
Drying	4-6h/90° C	Suggeste max moisture	0.15	%
Hopper	270÷280° C	Min temperture	270	° C
Front	280÷290° C	Max temperture	320	° C
Middle	280÷290° C	Injection rate	High	
Rear	295÷305° C	Injection pressure	40 ÷ 120	MPa
Nozzle	285÷300° C	Injection time	3 ÷ 15	Sec.
Hot Runner Temp,	295÷320° C	Screw Back	3,5	Bar
Moulds	80 - 120° C	Cooling time	30 ÷ 90	Sec.

Due to the high moisture absorption of PA6, special attention should be given to drying before processing. If the humidity exceeds 0.2%, it is recommended to dry in hot air at temperatures above 80° C for 8 hours. If the material has been exposed to the air for more than 8 hours, vacuum drying at 105° C for at least 8 hours is advised.

Melting Temperature: 280-300° C. For reinforced varieties, the melting temperature is 250-280° C.

Mold temperature significantly affects crystallinity, which, in turn, impacts the mechanical properties of the plastic parts. it is recommended to set the mold temperature at 80~90° C. For thin-walled, longer-flow plastic parts, such as the nylon cable tie production, a higher mold temperature is also recommended. Increasing the mold temperature can enhance the strength and rigidity of the plastic parts but reduces toughness.

Injection Pressure: Generally between 750-1250 bar (depending on the material and product design).

Injection Speed: High-speed (slightly reduced for reinforced materials).

Due to the short solidification time of PA6, the gate's position is crucial. The gate aperture should not be smaller than 0.5*t (where t is the thickness of the plastic part). If using a hot runner, the gate size should be slightly smaller than with a conventional runner, as the hot runner helps prevent premature solidification of the material. If using a submerged gate, the minimum diameter of the gate should be 0.75mm.