

MEXMID A GF30 H NC

Description	Polyamide 66 medium viscosity with 30% 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	1.36	g/cm ³	1183
Filler Content	30	%	3451
Relative viscosity (1% in 96% H ₂ SO ₄)	2.7 ± 0.10	-	307
Melting Point (DSC)	262	° C	3146

Mechanical Properties	Dry/Wet	Unit	ISO
Tensile elongation at break	3/5	MPa	527-2
Tensile strength at break	185/125	MPa	178
Flexural Modulus	8500/6500	MPa	178
IZOD Impact strength, notched (23° C)	11/19	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	Values	
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.

The information and data contained herein is believed to be accurate and are given in good faith. However, accuracy is not guaranteed and no warranty nor representation in such regard or in connection with any of the products referred to herein is given and PMexpolimeros disclaims all liability accordingly, whether in contract tort or otherwise.

