

Technical Support - Polyamide Product Literature -

1 Part Design

- » Introduction - the design checklist
- » Designing for Stiffness
- » Designing for the absorption of shock loads
- » Avoiding material concentrations and hot spots
- » Stress concentrations
- » Gating of a part
- » Weldlines

2 Tool Design

- » Flow of PA66
- » Cooling
- » Venting
- » Runners
- » Gating

3 Processing Guidelines

4 Troubleshooting Guide

- » Black specs
- » Part is not completely filled
- » Sink marks
- » Bubbles and voids
- » Flashing
- » Silver streaks
- » Cracking around ejector pin
- » Visible weld lines
- » Delamination
- » Bad surface finish
- » Brittle parts
- » Warpage and distortion
- » Jetting
- » Discolouration
- » Part sticks in tool

5 Electrical Properties

- » Ignition Testing
- » Insulative Properties
- » Thermal Properties
- » Arc Resistance

1 Part Design

» Introduction - the design checklist

Designing plastic parts is a complex process, however, a few general guidelines can be given. By following a structured approach when designing plastic parts, problem areas can be identified at an early stage. We generally recommend spending the necessary time to define requirements, before any design be made.

We recommend that you run through the following checklist before you start designing the part.

Based on the property profile and the functional analysis, it is usually possible to asses the commercial and technical feasibility of making a component in thermoplastic. At this point, a material selection is made, and you can start making the final component design.

After the part design has been finalized, usually a prototype is made via a stereo lithography or machining. Such prototypes can be used to check the looks of the part, to see whether or not it physically fits in place or to see if the motion transfer is carried out correctly. They should never be used to check mechanical performance.

Property Profile

- . 1.1 Target costs
 - 1.1.1 How many parts need to be made
 - 1.1.2 Estimated part volume
 - 1.1.3 Target cost for component
- 1.2 Environment
 - 1.2.1 Operating temperature range
 - 1.2.2 Expected service life @ operating temperature
 - 1.2.3 Peak temperatures
 - 1.2.4 Are parts loaded when peak temperatures occur
 - 1.2.5 List of chemicals in contact with part
 - 1.2.6 Is UV or weathering an issue
 - 1.2.7 Gamma, beta or other radiation
 - 1.2.8 Are there any abrasive media
- 1.3 Mechanical loads
 - 1.3.1 What are the abuse loads
 - 1.3.2 What are the continuous loads
 - 1.3.3 How high is the cyclic load, and how many cycles
 - 1.3.4 What maximum deformation is allowed
 - 1.3.5 What are the limits on the natural frequency of the part
 - 1.3.6 Any shock loads at minus temperatures
- 1.4 Wear and Friction
 - 1.4.1 In case of wear and friction, what is sliding velocity and the surface pressure
 - 1.4.2 How much wear is allowed
 - 1.4.3 Is wear debris a problem
 - 1.4.4 What is the external lubricating medium
 - 1.4.5 What is the maximum friction that is allowed
- 1.5 Electrical Properties
 - 1.5.1 Volume resistivity
 - 1.5.2 Dielectric strength
 - 1.5.3 Dielectric loss factor
 - 1.5.4 CTI value
 - 1.5.5 Arc resistance
 - 1.5.6 Ignition behaviour or flammability
 - 1.5.7 Smoke density in case of fire
 - 1.5.8 Smoke toxicity in case of fire
 - 1.5.9 Is build up of static electricity an issue

- 1.5.10 Is EMI shielding an issue
- 1.6 Pollution Control & waste disposal
 - 1.6.1 Is recycling an issue
 - 1.6.2 Are there unwanted chemicals (lead, PBB, PDBE, Chrom VI etc.)
- 1.7 Aesthetics
 - 1.7.1 Is the part visible
 - 1.7.2 What colours are needed
 - 1.7.3 Is the part matched with other coloured parts
 - 1.7.4 Surface requirements (glossy, dull etc.)
 - 1.7.5 Scratch resistance
- 1.8 Safety and standards
 - 1.8.1 Is this a safety critical part
 - 1.8.2 Are there any food standards involved (KTW, FDA, WRc, NSF etc.)
 - 1.8.3 Are there any electrical standards involved (UL, EDF, Kema, TÜV etc.)
 - 1.8.4 Any other standards

Functional analysis

- 2.1 Design space
 - 2.1.1 Is there room to add ribs
 - 2.1.2 Are there any surfaces where ribs can not be added for f.i. for aesthetic reasons
- 2.2 Kinematics
 - 2.2.1 What motion needs to be transferred
 - 2.2.2 What forced need to be transferred
- 2.3 Ergonomics
 - 2.3.1 Is a 'soft touch' needed
 - 2.3.2 Does the motion transfer need a certain 'feel'
 - 2.3.3 Is noise generation a problem
 - 2.3.4 Are sharp edges an issue
- 2.4 Environmental issues
 - 2.4.1 Are gaskets needed to keep out the environment (dust, oil etc.)
 - 2.4.2 Does the part need to keep out electro magnetic radiation
- 2.5 Tolerances
 - 2.5.1 Is thermal expansion an issue
 - 2.5.2 Is expansion due to moisture uptake an issue
 - 2.5.3 Are there any areas with specific tolerances (f.i. positioning pins, edges etc.)
 - 2.5.4 Flatness required
- 2.6 Heat conduction
 - 2.6.1 Does the part act as a heat insulator
 - 2.6.2 Does the part need to act as a heat sink

Conclusion

Based on the property profile and the functional analysis, it is usually possible to assess the commercial and technical feasibility of making a component in thermoplastic. At this point, a material selection is made, and you can start making the final component design.

After the part design has been finalized, usually a prototype is made via a stereo lithography or machining. Such prototypes can be used to check the looks of the part, to see whether or not it physically fits in place or to see if the motion transfer is carried out correctly. They should never be used to check mechanical performance

» Designing for Stiffness

Flexural loading of a beam

Compared to metals, plastics have low stiffness. However, the plastics allow far greater design freedom, so that using a ribbed design can compensate the lack of stiffness. The deflection of a beam under flexural load is inversely proportional with the product of the E-modulus and the moment of inertia of the cross section. In order to maintain equal stiffness in bending, the following should apply:

$$E[\text{metal}].I[\text{metal}] = E[\text{plastic}].I[\text{plastic}]$$

Since a wall thickness in excess of 4 mm should be avoided, just increasing the thickness of a part usually not feasible, and from a cost perspective not desirable. Figure 1 compares a EI of a 1.5 by 10 mm steel cross section to a rectangular cross section of equivalent stiffness in Polyamide 66, 30% GF, conditioned and to a U-section of equivalent stiffness in Polyamide 66, conditioned.

Steel: $E = 210,000 \text{ MPa}$
 $I = 1/12 * \text{Width} * \text{Height}^3 = (10 * 1.5^3)/12 = 33.75 \text{ mm}^4$
 $EI[\text{Steel}] = 210,000 * 33.75 = 7,087,500 \text{ N.mm}^2$

Equivalent stiffness on a rectangular cross section in PA66, conditioned:

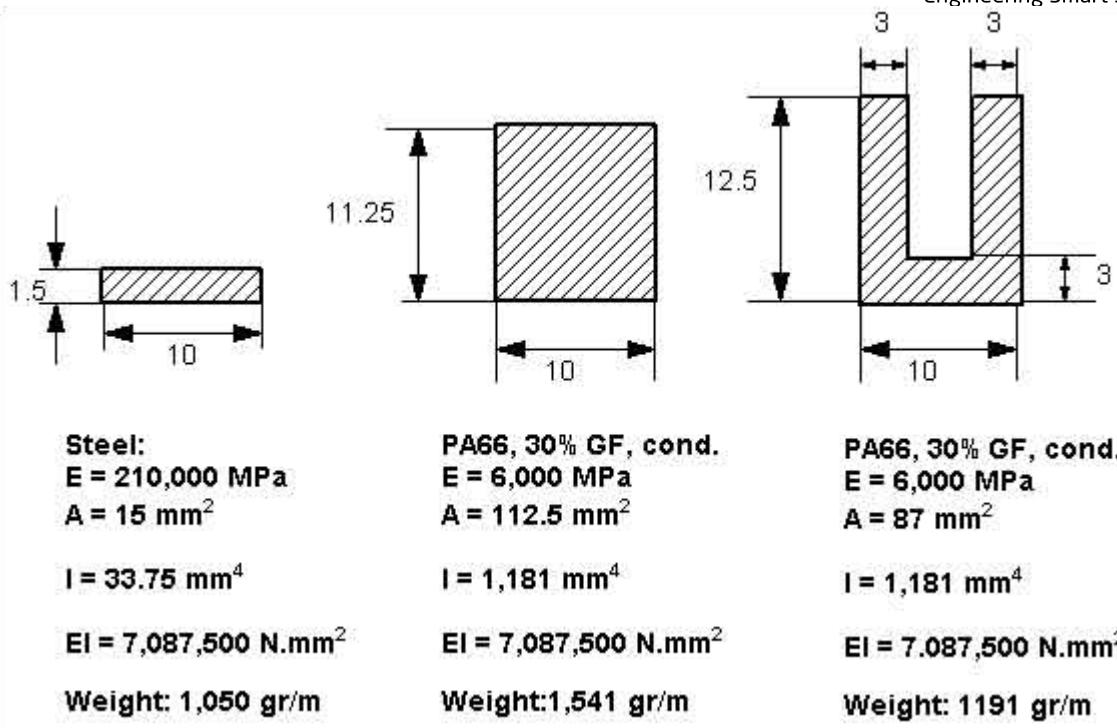
PA66, 30% GF: $E = 6000 \text{ N/mm}^2$ (Conditioned)
 $7,087,500 = 6000 * I[\text{PA66}]$
 $I[\text{PA66}] = 7,087,500/6000 = 1181 \text{ mm}^4 = 1/12 * \text{Width} * \text{Height}^3$
 Width = 10 mm, so
 $\text{Height}^3 = (12 * 1181)/10 = 1417 \text{ mm}^3$
 Height = 11.25 mm

Now the same for a U-section with a uniform 3 mm wall section:

PA66, 30% GF: $E = 6000 \text{ N/mm}^2$
 $7,087,500 = 6000 * I[\text{PA66}]$
 $I[\text{PA66}] = 7,087,500/6000 = 1181 \text{ mm}^4$

Using the appropriate formula, it follows that the height is 12.5 mm

This means the height is only 1.5 mm higher than a full cross section. At the same time, surface area of the cross section, directly proportional to the weight of the part, is 87 mm² instead of 112.5 mm²! Conclusion: use ribs whenever you can, both for cycle time and raw material usage.



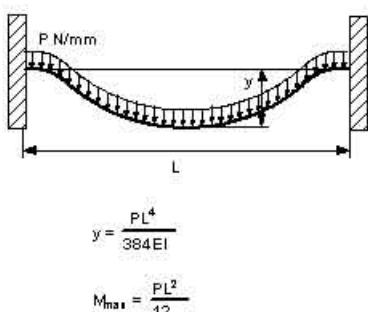
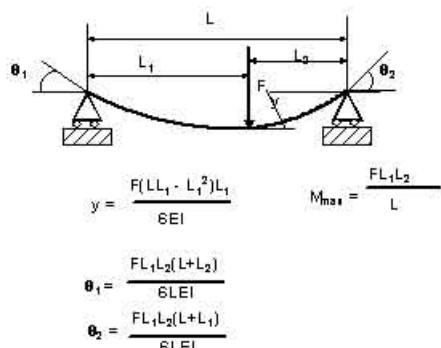
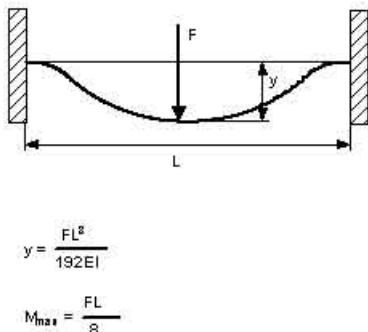
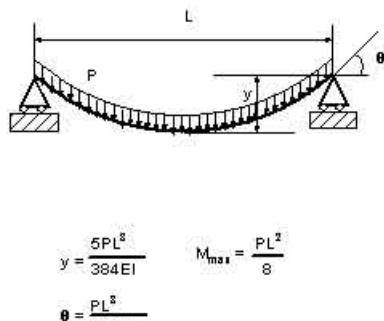
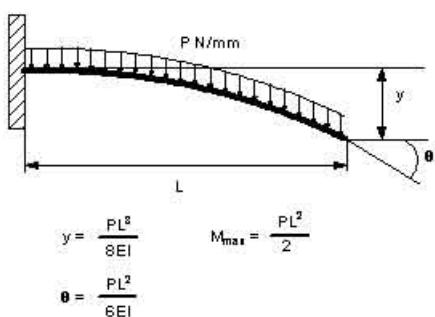
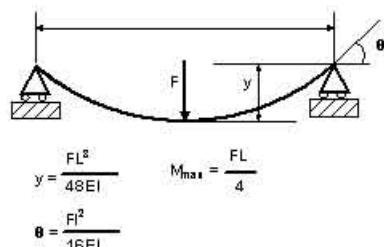
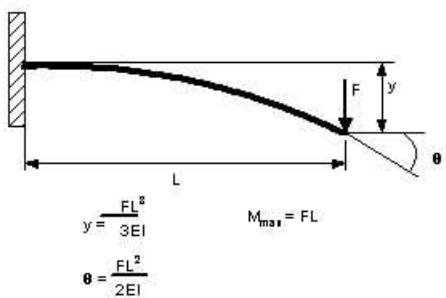
Deflection of beams

Designing for stiffness for a beam under flexural load is than quite simple: to get maximum stiffness, ensure you have a cross section with the highest possible moment of inertia around the axis of bending, within the following design rules:

- Maximum wall section should not exceed 4 mm
- Ribs should be thinner than the base surface to avoid sink marks
- Ensure ribs are rounded of at the base to avoid stress concentration
- Take measures for de-moulding, i.e a draft angle on the ribs of at least 1° and avoid long, thin cores

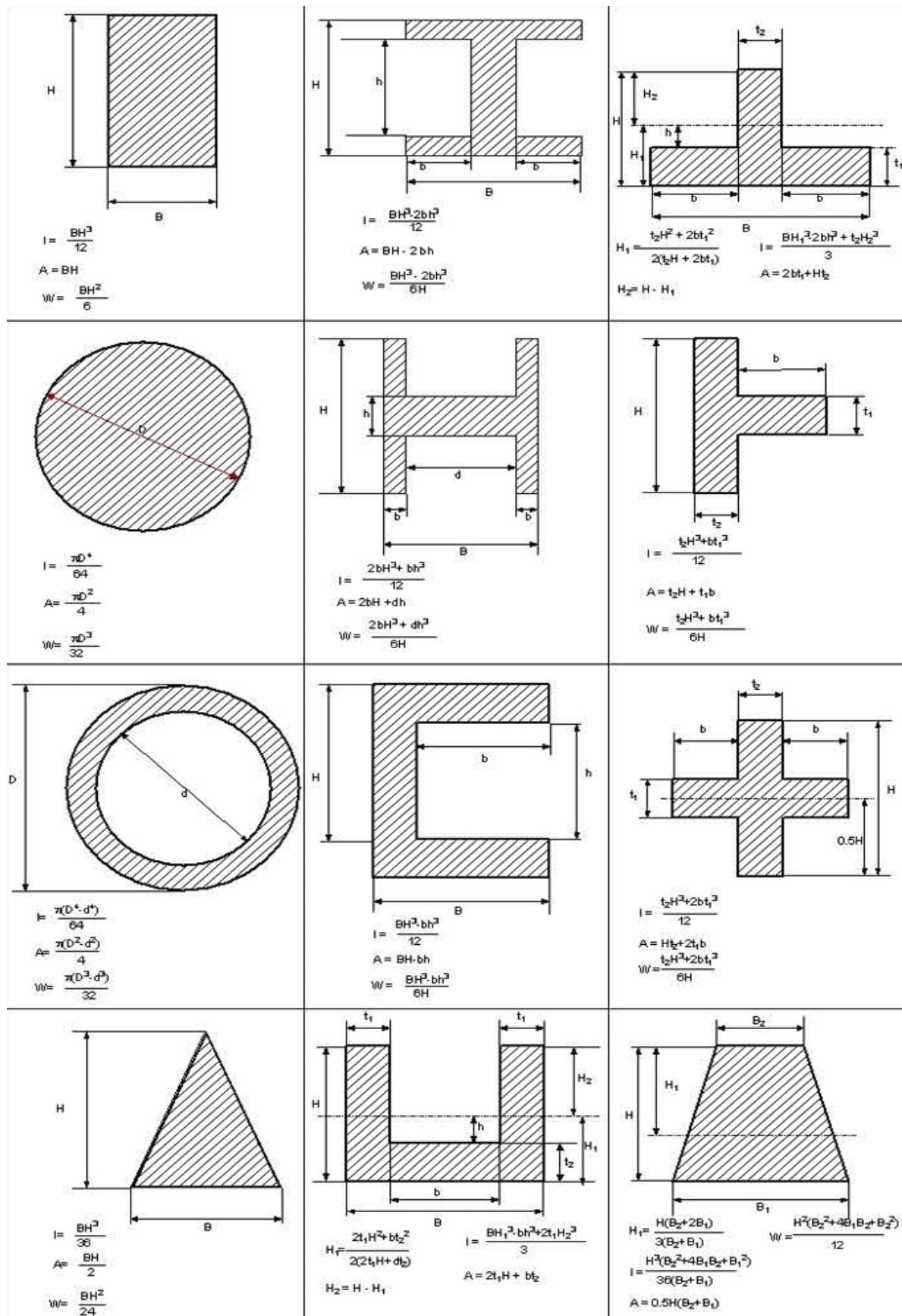
It also makes sense to ensure the cross section has a high moment of inertia in the areas where the bending load is highest. For instance, when a cantilever beam is loaded, the tip of the beam does need to have a particular high moment of inertia, however the end where it is fixed does need a high resistance to bending. Taken this into account saves cost, both in terms of material and cycle time, and saves weight.

The figure at the top give an overview of beams of a constant cross section behave under load.



Moments of inertia

The figure above shows how to calculate the moment of inertia for various cross sections.



Torsional loading of a beam

Torsional loads are more difficult to deal with in thermoplastic design. The basic formula for the angular twist of a beam is:

$$q = TL/KG$$

where q =angular twist in radians
 L = Length of the beam [mm]
 T =Torsion Moment [N.mm]
 K =Cross section dependend factor
 G =Shear modulus of a material [MPa]

For isotropic materials, the shear modulus G is given by the equation:

$$G = E/2(1+u)$$

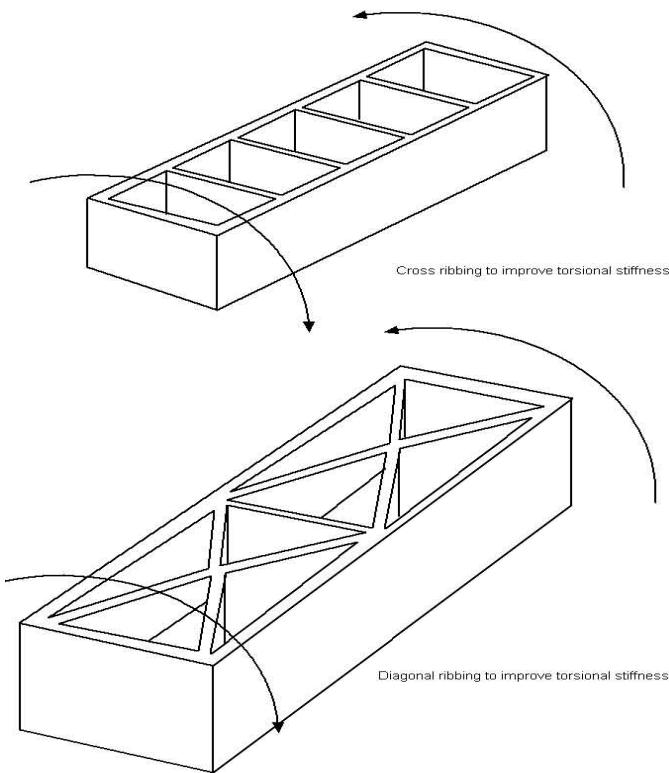
Where G = Shear modulus
 E =Tensile Modulus
 u =Poisson ratio = 0.25 to 0.35 for PA66

For unreinforced Polyamide 66, this formula gives a value close to reality. For reinforced thermoplastics, the shear modulus is lower than this value, as these materials are non-isotropic. Depending on how the orientation is compared to the fibre direction, G for Polyamide 66 GF 30 (conditioned) varies from 1200 to 2000 Mpa, whereas the value based on the formula of an isotropic material would lead you to expect a value of 2500 Mpa.

Compensating for the low shear modulus by using a cross section with a high K -value is also not easy. Closed cross sections give the highest K -value, but it is usually not possible to make closed cross sections with thermoplastics (only by using gas assist injection moulding in relatively simple geometries).

To maximize torsional stiffness, the following guidelines can be given:

- To maximize the K -value, concentrate as much material as possible in the centre of a cross section
- Use cross ribbing
- Use Diagonal ribbing



Tensile loading of a beam

Deformation under tensile loading is given quite simply by the formula:

$$\Delta l = F/A$$

Where Δl =change of length

F =applied force

A =surface area of the bar

To make the parts as stiff as possible, the surface area needs to be maximized, within the limits of good design. (Wall section not to exceed 4 mm)

Loading of flat plates

Formulas that approximate the deflection of flat plates are quite complex and involved. They can be used only for simple geometries:

- Uniform thickness which is thin compared to the dimensions of the plate
- Deflection less than one half of the plate thickness
- Isotropic, homogeneous material
- No ribbing, bosses etc. on the surface

For most situations, calculations with these formulas are only of limited use, and we recommend the use of finite element calculations for more accurate results. However, clearly putting diagonal ribbing on a large flat area will greatly enhance stiffness.

» Designing for the absorption of shock loads

Designing for the absorption of shock is quite complex. In each design, a different approach needs to be taken depending on the type of shock load, the function of the part, and the environmental conditions. In this section we can only give you a few general guidelines to help you on your way.

- The basis of design for shock absorption is to ensure your construction can deflect to absorb the load rather than providing a hard, rigid stop
- Distribute the elongation over the entire part In case of shock loads, a varying cross section will perform significantly better: thicker section where the bending moment is high, thinner sections where the bending moment is low. This will equalize the stresses throughout the part.
- Eliminate any abrupt stiffness changes in the construction This follows on from the previous item; anywhere where there is an abrupt change in stiffness, there will be a concentration of stress. These changes can be sudden changes in wall sections, ribbing, bosses, holes etc.
- Avoid stress concentrations and notches
- Ensure ribs are well rounded off at the base and use rounded corners for better stress transfer.
- Move weld lines out of high stress areas
- Move gates away from high stress areas
- Select the right material for the job

Polyamide 66 has better shock absorbing properties than most other engineering plastics. Impact modified grades are available. Before selecting a grade, please contact your Nilit representative who will help you select the right grade for your application

» Avoiding material concentrations and hot spots

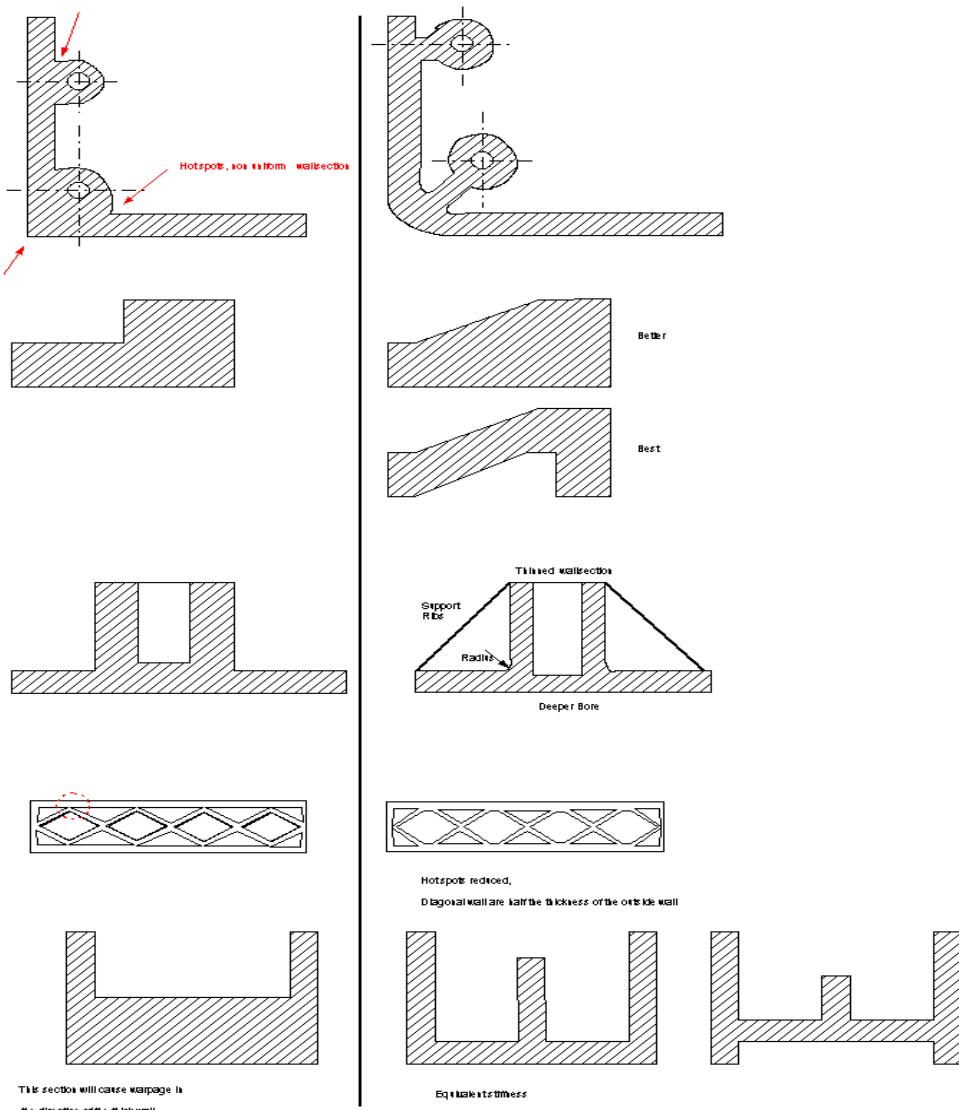
Material concentrations in whatever form should be avoided for a number of reasons:

- They result in abrupt stiffness changes, causing stress concentrations which can lead to premature failure of a part
- Often it is not possible to effectively pack a material concentration. This can lead to:
 - Internal stresses
 - Warpage of a component
 - Sink marks
 - In extreme cases to voids in the part.
- Cycle time will be longer than necessary, increasing the cost of the component

When designing a part, also consider that a component has to be cooled when it is moulded. Any area where there is no contact of the molten plastic with the tool will be a hot spot that results in increased cycle time.

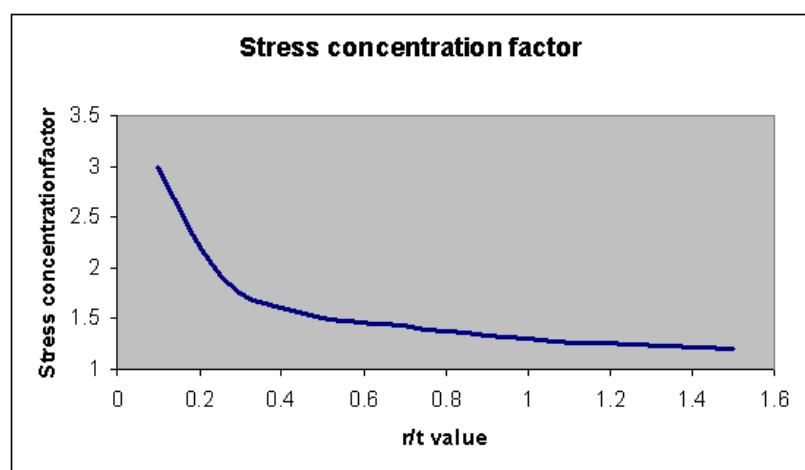
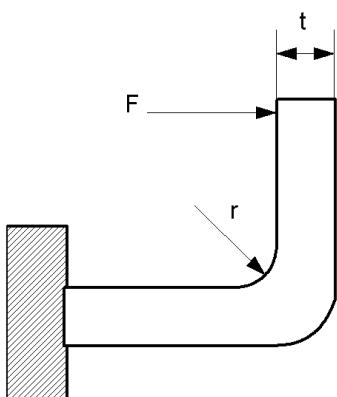
It is essential to ensure that the wall thickness is as uniform as possible. Thick walled sections will take longer to cool than thinner sections. As a result, the polymer chains in the thick part have more time to relax and crystallize, resulting in higher shrinkage than the adjoining thin parts. Thicker sections will continue to cool and shrink when thinner regions have already set, causing distortion and internal stresses.

Bear in mind that cycle time is directly proportional to the square of the wall thickness. Keeping the wall thickness down lowers raw material cost as well as processing cost!

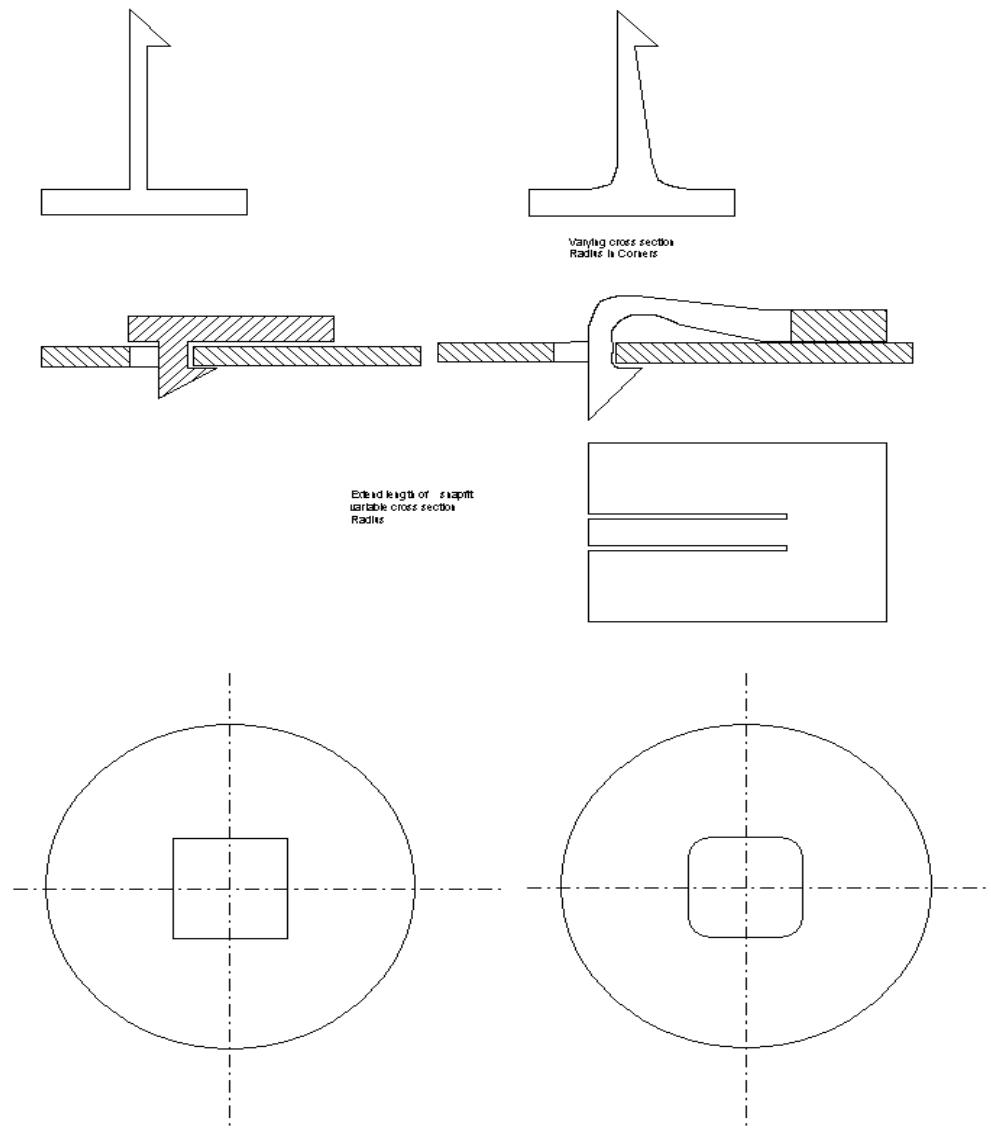


» Stress concentrations

Stress concentrations lead to premature failure of a part, particularly when subjected to shock loads. The Graph below shows the stress concentration factor as a function of the radius r/t of a transition in wall section.



Stress concentrations are caused by notches, such as radii that are too small, threads, self tapping screws, and also by any sudden change in stiffness of a part, such as ribbing, bosses, changes in wall section, inserts etc. In any of these cases, try and optimise the design of the part

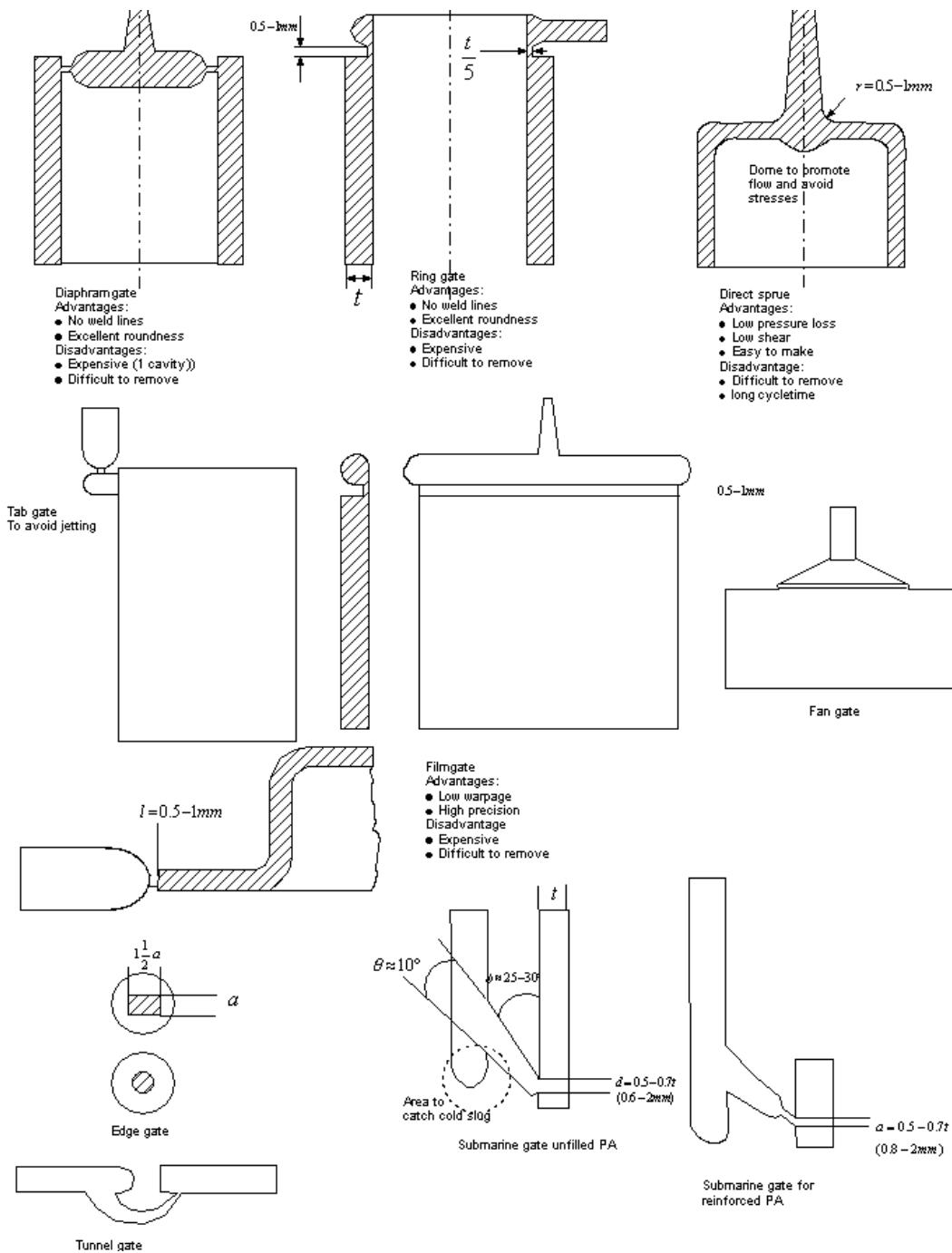


» Gating of a part

Gate location is a complex subject that involves many factors. Every part is different, so it is impossible to give exact rules on where and how to locate gating. Before selecting the type of gating use the following checklist:

- * What are tolerances on for instance flatness, roundness, parallel surfaces etc.
- * How critical is the mechanical performance, particularly on glass fibre reinforced thermoplastics
- * How large is the series
- * How cost critical is the component
- * What areas are aesthetically critical
- * Are there specific high load areas where you need to avoid weld lines
- * What is the general load direction

As with any design parameter, assembling these data before starting to make the design will make it easier to make the right choices, and will avoid expensive corrections in future.



Some common guidelines:

- Try to position the gate on the thickest section of a part. If this is not possible, consider re-design the thick section
- Do not inject straight into a thick section. This can cause jetting, resulting in bad surface finish and low mechanicals
- Avoid long flow paths
- Line up fibre direction with the general direction of the loads on the part
- Don't put gates in high stress areas

» Weldlines

Weld lines are a fact of life in thermoplastics; all we can do is move them out of harms way. They have two undesirable effects: they are visible and they lower the mechanical performance of a component.

Compared to many other thermoplastics, Polyamide 66 has excellent weld line strength. Unreinforced

materials retain over 90% of their original strength value, whereas reinforced grades drop to about 100 Mpa, independent of the glass fibre content. To ensure maximum weld line strength, the following factors should be taken into account:

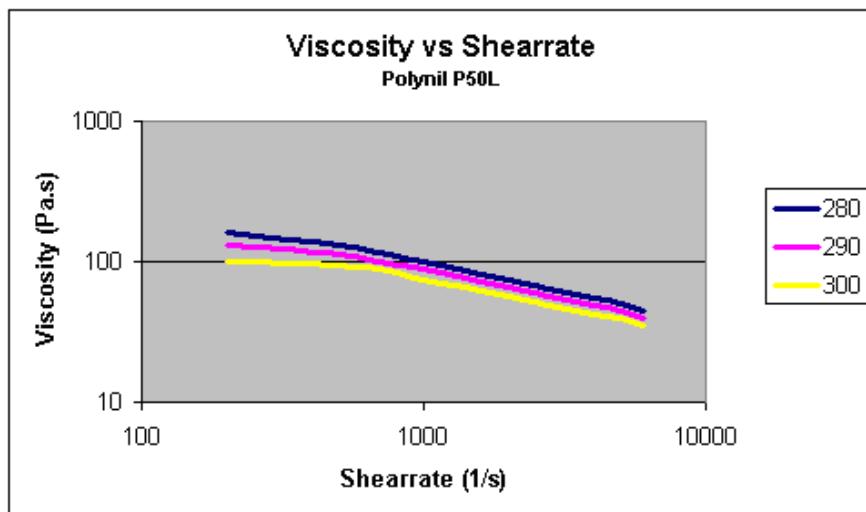
- Ensure excellent venting at the end of the weld line
- Maximize effective holding pressure; for instance make sure the weld line does not occur in a thin section far away from the injection point
- Ensure the melt is hot when the flow fronts meet

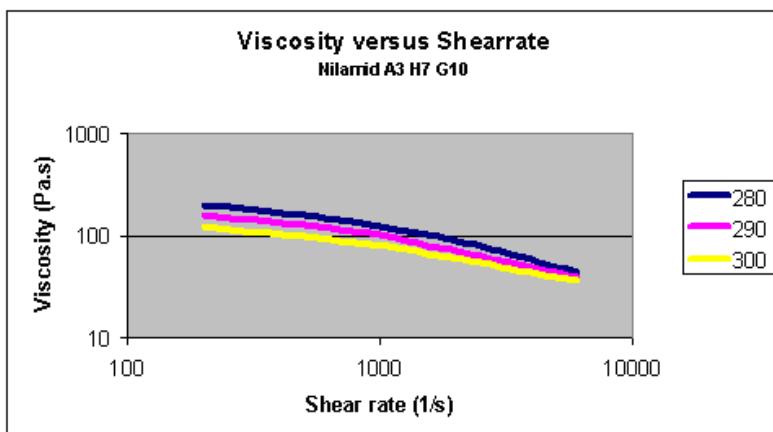
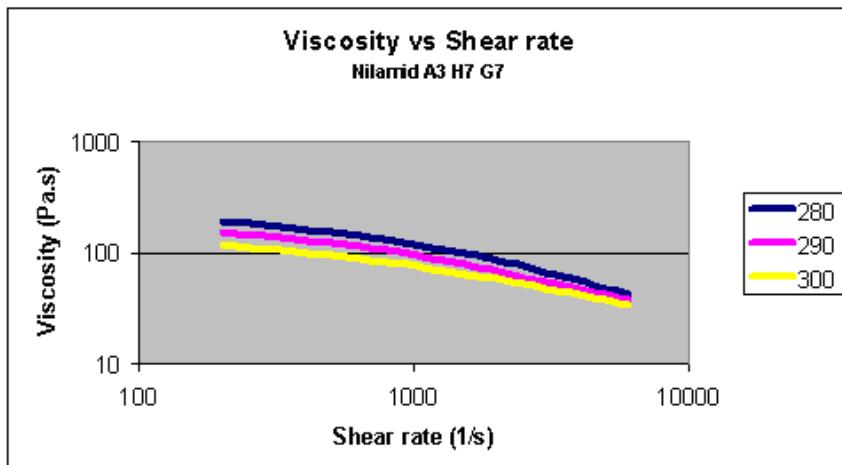
Again, with careful consideration and planning in advance the effect of weld lines can be minimized.

2. Tool Design

Flow of PA66

Polyamide 66 has excellent flow compared to many other engineering plastics, and it is possible to manufacture quite thin walled parts with these materials. The graphs below give the viscosity versus shear rate curves of Polynil P50L, Polynil P50FI, Nilamid A3 H2 G5, Nilamid A3 H2 G7 and Nilamid A3 H2 G10.





Cooling

The cooling system of a tool is an extremely important factor in the performance of the tool. Incorrect cooling can lead to both longer than necessary cycle times and to less than optimum product properties. The objective of good cooling is to maintain a constant temperature throughout the tool.

Properties affected by tool temperature include surface finish, dimensions (shrinkage and warpage) and mechanical performance (moulded in stresses). For PA66, the optimal tool temperature ranges from 80 °C to 120 °C.

Positioning of cooling channels is the key to good cooling. This positioning must be considered from the start when making a tool design. Too often, it is left too late in the design process, and channels are placed wherever there is room.

Some design tips for achieving good cooling:

- Take care in corners. Concave areas have a lower cooling rate than convex areas. It may be necessary to place an insert of a highly conductive material (e.g. Copper Beryllium) on the inside of a corner, or to put more cooling capacity on the inside.
- Ensure the flow in the cooling channels is turbulent, as this will give a 3 to 5 times better heat transfer than laminar flow. Channel diameters of 6 to 12 mm recommended.
- Ensure enough flow of the cooling medium to ensure the difference between the flow into the tool and the flow out of the tool is less than 2°C.

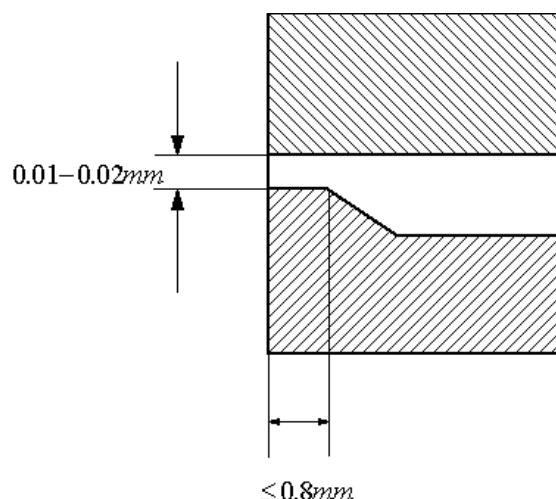
- Put cooling channels close enough together to minimize temperature fluctuation (usually 10 to 25 mm centre distance).
- Put cooling channels close to the cavity surface (15-25 mm from centre distance of the channel).
- Avoid long thin cores, which will be difficult to cool and will heat up during the moulding process.
- Ensure that cores are cooled by using heat pipes or spiral channels.
- Avoid hot spots in the product design

Venting

Mould cavities need to be vented to provide a way out for the air that is displaced by the material flowing into the tool. Poor venting will lead to low weld line strength, incomplete mouldings, burn marks and moulded in stresses.

Vents can be located anywhere along the parting line of the tool. They should be positioned where the tool fills up last. Vents should also be placed on bosses and blind ribs. Here, a flat spot on an ejector pin usually does the trick; otherwise inserts have to be used.

For Polyamides, the geometry of the vent is shown in the figure below. These vents will not give flashing, but are big enough to allow the air to escape.



Runners

Runners should convey the molten material to the tool with minimum loss of pressure. On the one hand, we want runners to have the minimum diameter, as this material needs to be either thrown away or re-processed, and larger diameters can mean longer cycle times. On the other hand, diameters need to be big enough to avoid pressure loss. To achieve this, a round cross section of the runner would be ideal, as it has the best surface to volume ratio. However, this is expensive to manufacture. A good compromise between pressure loss, manufacturing cost and de-moulding characteristics is usually found by using trapezoidal shape. Depth is about equal to base, and the taper is about 3-5 ° per side.

The runner lay out should be such that the pressure loss is from the sprue bushing to the gate is equal for each cavity. This can be achieved by ensuring the flow path is of equal length, or by varying the diameter of the runner. If the different cavities fill at different pressure, this will result in different dimensions, different levels of moulded in stresses and different mechanical performance.

It is recommended that at each runner intersection, the primary runner overruns the secondary runner by one diameter. This provides an area where colder, higher viscosity material can be trapped, allowing the hotter, lower viscosity material to flow into the tool.

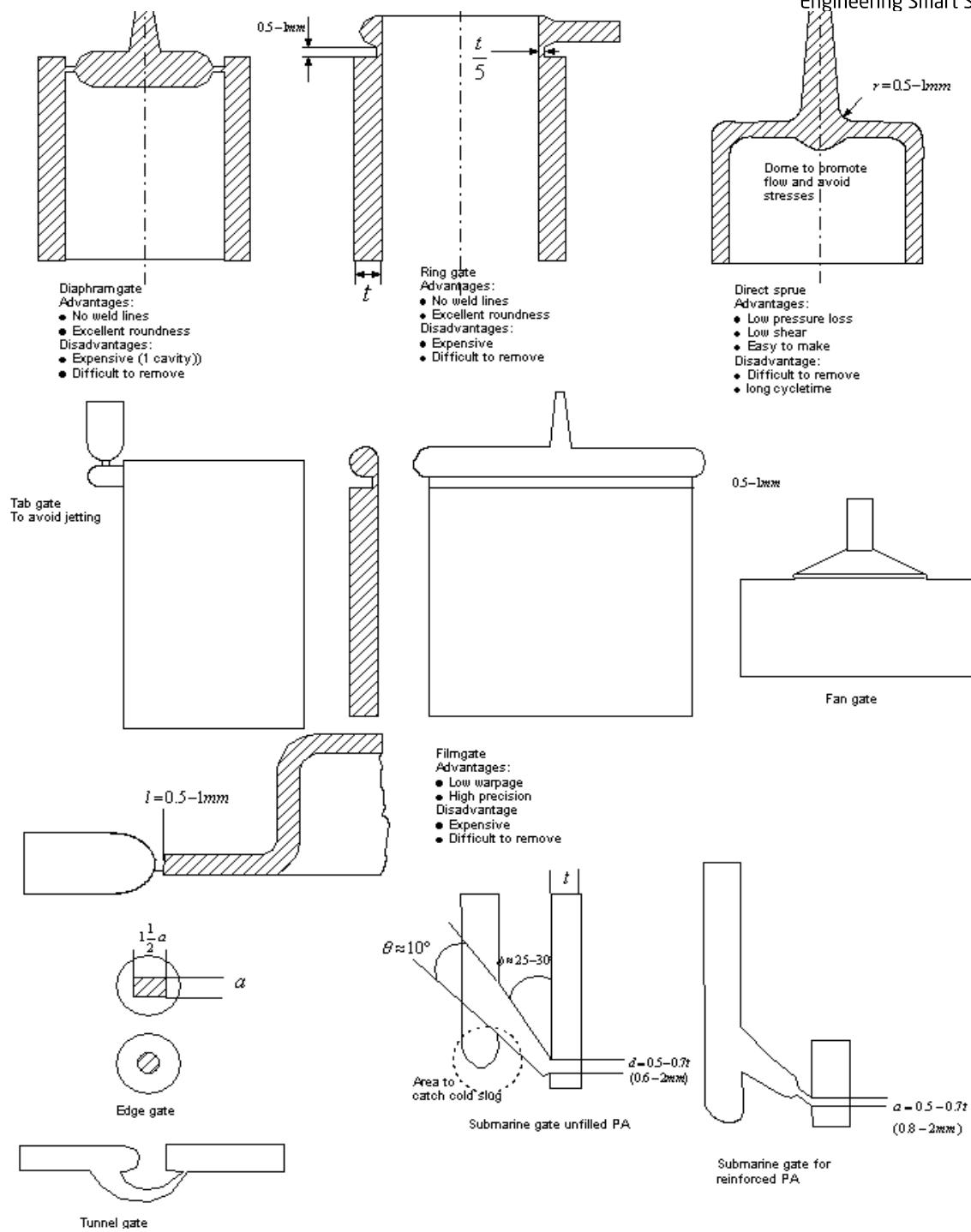
For high volume series, it is often attractive to use hot runners. For polyamide 66, it is recommended to use externally heated hot runners. Needle shut off valves can be used with these materials.

Gating

Gate location is a complex subject that involves many factors. Every part is different, so it is impossible to give exact rules on where and how to locate gating. Before selecting the type of gating use the following checklist:

- What are tolerances on for instance flatness, roundness, parallel surfaces etc.
- How critical is the mechanical performance, particularly on glass fibre reinforced thermoplastics
- How large is the series
- How cost critical is the component
- What areas are aesthetically critical
- Are there specific high load areas where you need to avoid weld lines
- What is the general load direction

As with any design parameter, assembling these data before starting to make the design will make it easier to make the right choices, and will avoid expensive corrections in future.



Some common guidelines:

- Try to position the gate on the thickest section of a part. If this is not possible, consider re-design of the thick section
- Do not inject straight into a thick section. This can cause jetting, resulting in bad surface finish and low mechanicals
- Avoid long flow paths
- Line up fibre direction with the general direction of the loads on the part
- Don't put gates in high stress areas

3. Processing Guidelines

This is a general overview for processing our products; injection moulding guides of specific products can be requested via mail or filling up the contact form in the technical support section of Nilit Plastics website.

Injection moulding of PA66

Polyamide 66 is easy to mould material, which is not particularly sensitive to moulding conditions. A few general guidelines are given here.

Pre-drying

Polyamide is hygroscopic and moisture sensitive, so pre-drying is recommended as a matter of rule. Material that is not pre-dried to a moisture level below 0.1% will degrade, causing surface defects, parts that are out of dimension and brittle parts. It is recommended to dry material for 4 hours at 80 °C to 85 °C in a desiccant dryer with more than one desiccant element.

A few tips to ensure proper operation of the dryer:

- Ensure the thermocouple that regulates the temperature is placed immediately before the entry of the air into the dryer. There can be a significant temperature drop in the air-conveyance system!
- The temperature of the air going out of the dryer silo should not be more than 30 °C lower than the air entering the system. If this is the case, you have insufficient air capacity.
- From time to time, monitor the dew point of the dry air to ensure the desiccant elements are functioning properly.
- Often, less air runs through the very bottom part of a dryer silo. Therefore, it is recommended that you take the material out of the bottom of the dryer and feed back into the top when you start up your process.

Moulding temperatures

For polyamide 66, the melt temperature must be kept below 300 °C. Any higher temperature will cause rapid degradation, which can be recognized by foaming of the material or splash marks on the surface of the part.

The following barrel settings are suggested as general conditions – please contact us about settings for specific products.

Material	Zone 1 (hopper)	Zone 2	Zone 3	Zone 4 (Nozzle)
Unfilled grades	260-295 °C	270-295 °C	275-290 °C	275-295 °C
FR Grades	260-280 °C	260-280 °C	270-280 °C	275-285 °C
Reinforced Grades	270-290 °C	270-295 °C	270-295 °C	275-295 °C
HI Grades	260-275 °C	260-280 °C	270-280 °C	275-285 °C

Tool temperature

Mould temperature is always a compromise. On the one hand, tool temperature should be as a high as possible to give optimum crystallization and dimensional, optimal surface finish and optimal mechanical performance. On the other hand, lower tool temperature can significantly cut cycle time. For Polyamide 66, 80 °C should be maintained as a minimum, for reinforced grades values of 90 to 110 °C are preferred.

Pressure and speed

Injection pressure should generally be around 70 to 100 Mpa; this results in a minimum clamping force of the moulding machine in tonnes of 0.7 times the projected surface area in cm².

Holding pressure is generally in the area of 90 Mpa.

For glassfibre reinforced compounds, the screw speed should be kept low, a rough indication is as follows:

Screw Diameter (mm)	Maximum rpm
20	150
30	100
40	70
50	60
60	50
70	40
80	35
>80	30

Back pressure should be kept to a practical minimum

Injection moulding of PA6

Polyamide 6 is easy to mould material, with a very wide processing window. A few general guidelines are given here.

Pre-drying

Polyamide is hygroscopic and moisture sensitive, so pre-drying is recommended as a matter of rule. Material that is not pre-dried to a moisture level below 0.1% will degrade, causing surface defects, parts that are out of dimension and brittle parts. It is recommended to dry material for 4 hours at 80 °C to 85 °C in a desiccant dryer with more than one desiccant element.

A few tips to ensure proper operation of the dryer:

- Ensure the thermocouple that regulates the temperature is placed immediately before the entry of the air into the dryer. There can be a significant temperature drop in the air-conveyance system!
- The temperature of the air going out of the dryer silo should not be more than 30 °C lower than the air entering the system. If this is the case, you have insufficient air capacity.
- From time to time, monitor the dew point of the dry air to ensure the desiccant elements are functioning properly.
- Often, less air runs through the very bottom part of a dryer silo. Therefore, it is recommended that you take the material out of the bottom of the dryer and feed back into the top when you start up your process.

Moulding temperatures

Polyamide 6 can be processed between 225 and 310 °C, depending on the grade used.

The following barrel settings are suggested as general conditions – please contact us about settings for specific products.

Material	Zone 1 (hopper)	Zone 2	Zone 3	Zone 4 (Nozzle)
Unfilled grades	220-260 °C	225-270 °C	225-270 °C	225-275 °C
FR Grades	225-260 °C	230-260 °C	235-265 °C	235-265 °C
Reinforced Grades	240-280 °C	240-290 °C	240-290 °C	240-295 °C
HI Grades	220-265 °C	225-260 °C	225-265 °C	230-275 °C

Tool temperature

Mould temperature is always a compromise. On the one hand, tool temperature should be as high as possible to give optimum crystallization and dimensional, optimal surface finish and optimal mechanical performance. On the other hand, lower tool temperature can significantly cut cycle time. For Polyamide 6, 80 °C should be maintained as a minimum, for reinforced grades values of 90 to 110 °C are preferred.

Pressure and speed

Injection pressure should generally be around 70 to 100 Mpa; this results in a minimum clamping force of the moulding machine in tonnes of 0.7 times the projected surface area in cm².

Holding pressure is generally in the area of 90 Mpa.

For glassfibre reinforced compounds, the screw speed should be kept low, a rough indication is as follows:

Screw Diameter (mm)	Maximum rpm
20	150
30	100
40	70
50	60
60	50
70	40
80	35
>80	30

Back pressure should be kept to a practical minimum

4. Troubleshooting Guide

Black specs

Black spec range from small black particles to big lumps of black material. Most often, the cause is contaminated machinery, e.g. drier, moulding machine or hot runner.

Design related causes None

Tool related causes

1. Dead spot in hot runner Solution: Re-design

Process related causes

1. Machine not cleaned properly 2. Contaminated regrind 3. Worn out shut off valve 4. Dryer not clean	Solution: Purge with purging compound Solution: Use clean regrind Solution: replace Solution: Clean dryer & transport system
----------------------------------------------------------------------------------------------------------------	---------------------------------------------------------------------------------------------------------------------------------------

Part is not completely filled

When the filling is irregular, this is usually related to variations in the material or to defects on the machine. If the filling problem is evident on all moulded parts however, it is usually related to the process or the tool design.

Design related causes

1. Gate too small 2. Wall section too thin 3. Thick ribs or bosses on a thin surface 4. Too much variation in wall section	Solution: increase size Solution: use higher flow material or increase wall section Solution: change wall section Solution: redesign
-------------------------------------------------------------------------------------------------------------------------------------	-----------------------------------------------------------------------------------------------------------------------------------------------

Tool related causes

1. Runner too long or too thin 2. Trapped air 3. Unbalanced runner system 4. Wrong gate position 5. Too sharp corners (pressure drop) 6. Gate freezes of too fast	Solution: increase runner size Solution: improve venting Solution: balance runner system Solution: Change gate position Solution: Round off corners Solution: Increase gate size
----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------	-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------

Process related causes

1. Melt temperature too low 2. Not enough injected volume 3. Tool temperature too low 4. Injection pressure too low 5. Injection speed too low 6. Inconsistent plasticizing in barrel 7. Worn out shut off valve 8. Nozzle blocked with burned material	Solution: check heater bands or increase nozzle temperature Solution: increase volume Solution: increase tool temperature Solution: increase injection pressure Solution: increase Solution: check barrel wear or use machine with larger barrel Solution: replace Solution: clean
------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------	---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------

Sink marks

Sink marks is one of the most common problems with thermoplastics, and be caused be either part design, tool design or process conditions.

Design related causes

1. Thick ribs or bosses on a thin surface 2. Too much variation in wall section	Solution: Change wall section Solution: even out wall section
------------------------------------------------------------------------------------	------------------------------------------------------------------

Tool related causes

1. Too sharp corners (high pressure drop)	Solution: round off corners
2. Too small or too long runners	Solution: increase runner size
3. Gate freezes off too fast	Solution: increase gate size
4. unbalanced runners system	Solution: balance runner system

Process related causes

1. Holding pressure too low	Solution: increase
2. Holding time too short	Solution: increase
3. Ejection temperature too high	Solution: increase cooling time
4. Tool temperature too low	Solution: increase
5. Injection volume too low	Solution: increase

Bubbles and voids

Voids are mostly caused by too thick wall sections. Bubbles are mostly due to air that is entrapped in the screw.

Design related causes

1. Wall section too thick	Solution: core our wall or use higher viscosity material
---------------------------	----------------------------------------------------------

Tool related causes

Voids are usually not tool related

Process related causes

1. Excessive barrel or screw wear	Solution: check and replace
2. Wet material	Solution: pre.dry
3. Thermal degradation	Solution: lower melt temperature or residence time

Flashing

Flashing and short shots are two sides of the same coin, and often caused by similar problems.

Design related causes

Flashing is usually not caused by the design of the part

Tool related causes

1. Unbalanced runner system	Solution: balance runner system
2. Surface finish on tool plates too rough	Solution: polish
3. Vents too large	Solution: put in new inserts

Process related causes

1. Insufficient clamping force	Solution: use machine with more clamping force
2. Excessive injection or holding pressure	Solution: decrease
3. Too high melt temperature	Solution: decrease

Silver streaks

Wet material is the most common cause of silver streaks; for PA66 an overheated melt also causes this effect.

Design related causes

Silver streaks are usually not caused by the design of a part

Tool related causes

1. Cold slug in part	Solution: increase area to catch cold slug
----------------------	--------------------------------------------

Process related causes

1. Wet material	Solution: predry
2. PA66: Melt over 300°C	Solution: reduce melt temperature or residence time
3. Too much lubricant or release agent	Solution: reduce

Cracking around ejector pin

This effect is usually not design related

Tool related causes

1. Ejector pins too small or placed incorrectly	Solution: correct
2. Vacuum is formed under part	Solution: apply air ejection

Process related causes

1. Overpacking	Solution: reduce holding pressure
2. Ejection temperature too high	Solution: increase cooling time or use nucleated material

Visible weld lines

Although some guidelines can be given to improve the situation, it can sometimes be very difficult to get rid of visible weld lines.

Design related causes

Tool related causes

1. Gate positioning	Solution: move gate
2. Trapped air	Solution: improve venting
3. Runner system too long	Solution: increase cross section

Process related causes

1. Injection or holding pressure too low	Solution: increase
2. Pigment distribution	Solution: use other pigments or masterbatch

Delamination

Delamination most often occurs when moulding polymer blends

Design related causes

1. Too high shear rates	Solution: increase wall sections or round off sharp corners
-------------------------	-------------------------------------------------------------

Tool related causes

1. Too much shear	Solution: increase gate sizes
-------------------	-------------------------------

Process related causes

1. Contamination with other material	Solution: check dryer and purge machine with cleaning compound
--------------------------------------	----------------------------------------------------------------

Bad surface finish

This is one of the most common problems; often it stems from tool temperatures below manufacturers recommendations.

Design related causes

1. Large steps in wall section causing irregular filling	Solution: even out wall sections
2. Material stalls in certain places during filling	Solution: optimize part geometry

Tool related causes

1. Runner system too small	Solution: increase cross section of runner
2. Gates too small	Solution: increase size
3. Flow length too long	Solution: use lower viscosity material

Process related causes

1. Tool temperature too low	Solution: increase
2. Injection speed too low	Solution: increase

Brittle parts

Brittle parts are usually related to degraded material, either as a result of a high moisture level or as a result of excessive temperatures and residence times.

Design related cause

1. Stress concentrations	Solution: round off sharp corners
2. Thin wall section causing excessive shear	Solution: increase wall section or use higher flowing material

Tool related causes

1. Weldlines in highly stressed areas	Solution: re-position gating
2. Small gating causing excessive shear	Solution: increase gate sizes

Process related causes

1. Material degradation by moisture	Solution: pre-dry material
2. Material degradation through high melt temperature	Solution: reduce melt temperature
3. Material degradation through long residence times	Solution: reduce machine size
4. Melt temperature too low temperature	Solution: check heater bands, increase
5. Moulded in stresses temperature	Solution: increase tool temperature and melt

Warpage and distortion

Warpage is mostly caused by unfavourable fibre orientation. Here it is important to find the correct way to gate a part, and position gates in the right position.

Design related causes

1. Shrinkage causes part to distort or amorphous base polymer)	Solution use a low shrinkage material (high filler level
2. Ribs place incorrectly	Solution: adjust wall sections or rib locations
3. Too much variation in wall sections	Solution: even out wall sections

Tool related causes

1. Unequal tool temperature distribution	Solution: improve cooling
2. Parts stick in tool upon ejection	Solution: check draft angles and ejector pin location
3. Unfavourable flow direction	Solution: Check gate position

Process related causes

1. Parts stick in tool due to overpacking	Solution adjust holding pressure
2. Injection speed too low	Solution: increase
3. Tool temperature too low	Solution: increase

Jetting

Jetting occurs whenever a gate is located in a position where it injects into a large open space without resistance

Design related causes

Jetting is not related to the design of the part as such

Tool related causes

1. Gate in wrong position flow	Solution: re-locate gate to increase resistance to polymer
--------------------------------	------------------------------------------------------------

Process related causes

Jetting is not related to processing conditions

Discolouration

Discolouration can be caused by excessive dryer temperatures, but also by degraded material.

Design related causes

Discolouration is not related to the design of the part

Tool related causes

Discolouration is not related to the tooling.

Process related causes

1. Dryer temperature too high
2. Too much regrind
3. Contamination
4. Residence time too long
5. Too much de-compression

Solution: Check dryer temperature against Nilit recommendation
Solution: check level and quality of regrind
Solution: check dryer and feed throat for contamination
Solution: check part weight versus machine size
Solution: check

Part sticks in tool

Design related causes

This effect is usually not related to the part design

Tool related causes

1. Undercuts and draft angles

Solution: check and correct

Process related causes

1. Cooling time too short material
2. Too high holding pressure
3. High injection pressure needed to fill part

Solution: increase cooling time or switch to nucleated
Solution: check and correct
Solution: use lower viscous material

5. Electrical Properties

Ignition Testing

There are many different national and international regulations that apply to the ignition and burning behaviour of thermoplastics. Some of the most important are explained briefly here. For each market and region, specific legislation should be checked.

UL 94 Flammability testing

Underwriters Laboratories UL 94 ratings are the most widely accepted standards for flammability performance of thermoplastics. They are intended to determine a material's ability to extinguish a flame. Materials are classified based on rate of burning, time to extinguish, ability to resist dripping and whether or not drips are burning. The main tests for this classification are UL 94 HB, UL 94 V2/V1/V0 and UL 94 5 VB, listed in order of severity of the test. The rating of a material is affected by its colour and the thickness of the test bar. Therefore the UL rating should apply to the thinnest wall section in the actual plastic part.

For our UL Listings, look at the following page: http://www.nilit.com/plastics/certifications_ul-listings.asp

UL 94 Hot Wire Ignition testing

The purpose of this test is to determine the materials ability to resist ignition when it comes into contact with glowing elements, like overheated electrical wires or contacts. For this test, a copper wire is wrapped around a sample, and heated up 930 °C. The time it takes to ignite a sample is recorded. UL divides up the HWI ratings into the following performance categories:

Ignition time	UL PLC number
Less than 7 seconds	5
7 to 15 seconds	4
15 to 30 seconds	3
30 to 60 seconds	2
60 to 120 seconds	1
Over 120 seconds	0

HWI Listings for various grades:

IEC 695-1-1 Glowwire test

The purpose of the glow wire test is asses the fire risk when a component in the vicinity of thermoplastic parts overheat. A glow wire with a defined temperature is brought into contact with the thermoplastic for 30 seconds three successive times. Two different temperature are defined:

GWIT:

The highest temperature where there is no ignition of the plate after 3 successive contacts with the glow wire, with 25 °C added on to the temperature. The value is dependent on the wall thickness, so the temperature is always accompanied by a wall thickness.

GWFI

The highest temperature at which the flame extinguishes within 30 seconds after each of the 3 applications. This value is also depends on the wall thickness.

The following Nilamid grades have a UL yellow card listing:

Limiting Oxygen Index

The limiting oxygen index is defined as the minimum concentration of oxygen in the atmosphere, expressed as a percentage by volume, which can sustain flame on a thermoplastic material. The test is carried out in an externally controlled atmosphere of nitrogen and oxygen. Materials with LOI values over 21% will not burn under atmospheric conditions.

The LOI value of PA66 in it's natural form is 28%; flame retardant grades have values of up to 35% LOI

Insulative Properties

Many insulative properties of polyamides depend on the moisture content and temperature of the material. For this reason, actual operating conditions should always be taken into account. As with flame retardancy, many different, local test standards are available. Here is an overview of the major international tests.

Volume resistivity

The volume resistivity shows the resistance of a material to the flow of current. It can be measured according to different test standards; the values mentioned here are measured according to ISO 1325. Graph 1 shows the volume resistivity of Polynil P50L versus temperature, whereas graph 2 illustrates the effect of moisture.

Comparative Tracking Index (CTI)

The comparative tracking index (IEC 112 or UL746) is carried to test the resistance of plastics against superficial tracking currents under humid conditions. The maximum voltage that can be measured in this test is 600V; Polyamides exhibit excellent CTI values.

On the UL card, the PLC's are given to the the CTI:

>600 V: PLC 0
 400-600 V: PLC 1
 250-400V: PLC 2
 175-250V: PLC 3
 100-175 V: PLC 4
 <100V: PLC 5

Dielectric strength

The dielectric strength is the voltage at which the insulative properties of a material are destroyed. An arc passes through the material at this point. The dielectric strength is depended on temperature, moisture content and wall thickness. The graph below illustrates the effect of temperature

Thermal Properties

Heat resistance is divided up into short-term heat resistance, i.e. the ability of the material to withstand peak temperatures, and long-term heat aging. The latter indicates the ability of the material to withstand loss of properties as a result of thermo-oxidative degradation.

Heat Deflection temperature

The Heat Deflection Temperature Under Load (HDTUL) gives an indication if the materials ability to withstand peak temperatures under load. It is measured by putting a flexural load of 1.8 or 0.45 MPa on a test bar, and than heating up the bar at a controlled rate of temperature increase (2°C per minute). The HDTUL is defined as the temperature where the deflection exceeds a specified value (about 0.3 mm).

Relative Temperature Index

The Relative Temperature Index (RTI) as defined in UL 746 gives an indication of the resistance of a material to long-term heat aging. The RTI is given for three categories: electrical, mechanical with impact and mechanical without impact. The RTI is defined as the temperature at which, after an exposure of 60,000 hours, a property has dropped of to 50% of its original value. For the electrical listing, usually the dielectric strength is chosen as the test parameter, for mechanical with impact usually the Izod impact strength or the tensile impact test, and for mechanical without impact usually the tensile strength is taken as the reference. The RTI depends on the wall thickness; lower wall thickness reduces the RTI.

To ensure that the results are not influenced by temperature fluctuations of the oven, a reference material with known RTI values is tested alongside the trial material. The RTI is measured relative to the reference material, hence the name Relative Temperature Index. RTI listing is extremely expensive, and is not carried out for most materials. On the UL yellow cards, many materials carry only the intrinsic performance listing of a base polymer. In the table below both the actual listed value of our products is mentioned as well as an estimated real value

Arc Resistance

For the arc resistance, two different situations can occur: high Voltage, low current arcing and Low voltage, high current arcing. UL 746 has defined a test for each situation: High Voltage Tracking Resistance (HVTR), High Amperage Arc Ignition (HAI) and the high Voltage, dry arc resistance.

High Voltage Tracking Resistance (HVTR)

This test determines how well a material resists forming a conductive path on the surface when it is subjected to repeated high voltage arcing. To this effect, two electrodes charged with 5200 V are brought on the surface of the part. As soon as an arc forms, the electrodes are separated. This process is repeated for two minutes. At the end of the test, the length of the conductive path is measured and the tracking rate (in mm/min) is determined. UL recognizes the following classes:

High Amperage Arc Ignition (HAI)

For this test, two electrodes are placed on the surface and charged with 240 V (60 Hz): they are moved closer together until an arc forms on the surface. When this happens, the electrodes are separated and the process is repeated. This happens at a rate of 40 arcs per second. This test is carried out at a far lower voltage, but far higher amps than the HVTR test. The HAI is defined as the number of arc needed to ignite a sample. Polyamide 66 has excellent resistance against this type of arcing. With a resistance of over 120 arcs, it falls into the highest performance class defined by UL.

High Voltage Dry Arc Resistance

This test determines how well a material resists forming a conductive path on the surface when it is subjected to a continuous arc. This test is described in ASTM D495. Two electrodes are applied to the surface at a specified distance. According to a defined cycle, the electrodes are charged with a Voltage of 15000 V. The time to failure (either through ignition of the part or by a current that starts to run between the electrodes) is noted. Polyamide 66 does not have a very good resistance to this type of arcing; failure times are between 60 and 120 seconds.