Reaction Injection Molding

DESIGN GUIDE

Bigger, Stronger, Lighter™

ROMEORIM
REACTION INJECTION MOLDING

74000 Van Dyke, Romeo, MI 48065 · 586-336-5800 · romeorim.com

What Drives Us?

Why?

We believe in serving those who want to make a difference.

What?

The way we make a difference is through
- Servant Leadership Culture
- Relationships & Accomplishments
- Innovative solutions
- Continuous Improvement

How?

We provide bigger, stronger and lighter solutions that our customers need to be successful

The information contained in this Design Guide is for general information purposes only. The information is provided by Romeo RIM and Romeo RIM commits to keep the information up to-date and correct, we make no representations or warranties of any kind, express or implied, about the completeness, accuracy, reliability, suitability or availability with respect to the Design Guide or the information, products, services, or related graphics contained in the document for any purpose. Any reliance you place on such information is therefore strictly at your own risk. In no event will Romeo RIM be liable for any loss or damage including without limitation, indirect or consequential loss or damage, or any loss or damage whatsoever arising from loss of data or profits arising out of, or in connection with, the use of this Guide. Romeo RIM will make every effort to keep the Design Guide updated to the last advances in technology. However, Romeo RIM takes no responsibility for, and will not be liable for, the notifications or publishments of new technology and information. Romeo RIM believes the information and recommendations herein to be accurate and reliable as of the date printed and that Romeo RIM's products mentioned herein are reasonably fit for the recommended purposes.

Romeo RIM Design Guide
Table of Contents

Chapter 1 – Introduction to Reaction Injection Molding (RIM)

1.1 The RIM Process

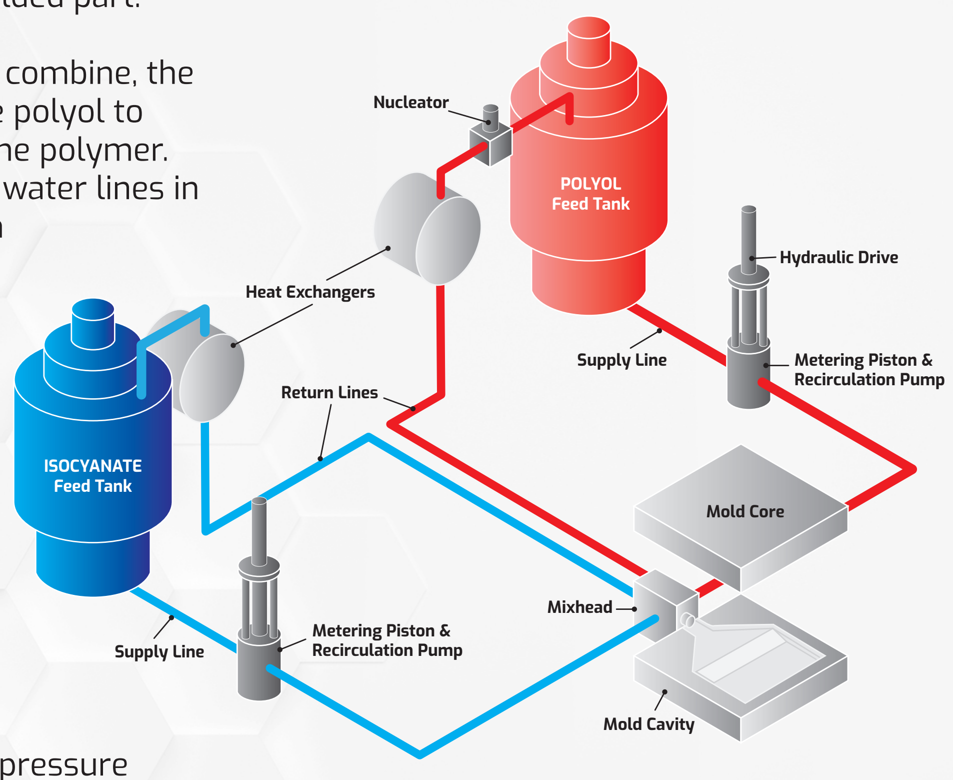
Reaction Injection Molding (RIM) is the process by which two liquid components chemically react to form a plastic part. The two liquids are isocyanate (iso) and polyol. Iso is referred to as the A side component and is the catalyst. Polyol is referred to as the B side and determines the physical characteristics of the molded part. Density, impact strength, flex modulus, color, and other properties can all be influenced by additives in the polyol.

Mixing additives to the polyol such as stabilizers, flow modifiers, combustion modifiers, filler, blowing agents, catalysts, pigments, and release agents control the physical characteristics of the molded part.

When the “A” and “B” components combine, the iso reacts with the hydroxyl in the polyol to form a thermosetting polyurethane polymer. This is an exothermic reaction, so water lines in the mold are required to maintain proper molding temperatures.

The diagram below shows a typical RIM material system. The pressurized day tanks can hold from 30 gallons to 250 gallons of iso or polyol depending on the size of the system. Recirculation pumps and agitators maintain a homogeneous blend of the individual components. Heat exchangers are used to maintain temperature in the system. High pressure pumps or cylinders meter the iso and polyol into the mix head. Flow rates, pressures, and temperatures are controlled to achieve quality molded parts.

The mix head contains injector nozzles which impinge the iso and polyol at ultra-high velocity to provide excellent mixing. The velocity of the material transfers from turbulent to laminar flow when passing the dam gate. This laminar flow in conjunction with the vents in the tools allow the part to fill properly.



The Advantages of RIM Process:

Compared to thermoplastic injection molding, which requires high heat and high pressure to press melted plastic pellets into a steel mold, RIM parts are formed when two liquid components (polyol and isocyanate) chemically react inside a mold.

The RIM process offers many advantages over competing technologies including:

Very Large, Lightweight parts

The “flowability” of polyurethane components means the material is distributed evenly inside the mold. This lets you produce large parts, which is not possible with injection molding. Because mold pressures are much lower, large presses are not necessary.

Varied Wall Thickness

Producing significant variable wall thickness within the same molded part is achievable with RIM. Cross sectional wall thickness ranges between .25 inches and 1.125 inches are possible in the same molded part. This is NOT true for thermoplastic injection molding, blow molding, sheet molding compound (SMC) and other polymers

Low Tooling Costs

RIM tooling costs are significantly less than that of an injection mold. Because the RIM process incorporates low mold temperatures and pressure, the tool can be made out of less expensive mold materials other than steel such as cast aluminum, aluminum, kirksite alloys, nickel, epoxy, silicone and fiberglass. The choice depends on such factors as the number of parts to be made; dimensions, shapes, and tolerances; the quality and texture of the surface; mold life; required mold cost; and part performance. The larger the mold, the greater the savings.

Freedom of design

RIM lets you mold highly detailed, intricate parts at relatively low tooling and capital equipment costs. Parts with varying wall thicknesses can be designed into the same molded part.

Encapsulation of Inserts

Different types of inserts can be placed into a mold prior to injection of the RIM material, allowing the RIM material to encapsulate the insert into the part. Inserts such as steel, aluminum, glass, wood, electronic sensors, PC boards and wiring harnesses are some examples of material that have been encapsulated.

Rapid Prototypes

Excellent working prototypes can be developed with lead times of 3-15 days, at a cost much less than traditional injection molding.

Class A Surfaces

The surface finish of RIM parts allows manufacturers to produce Class A painted parts capable of producing high-gloss finishes that match high-gloss painted metal parts.

In-Mold Painting

With the RIM process, it’s possible to apply gel-coats and two-component polyurethane in-mold paints into the mold prior to injection. The injected polyurethane material bonds to the gel-coat or paint during molding, allowing a decorated part to be produced in the mold. This can significantly reduce secondary finishing costs.

Other Benefits

High strength, dimensional stability, good weatherability, scratch resistant, heat resistant, impact resistant, resistant to organic and inorganic acids, and high R Value.

Chapter 2 – Part Design

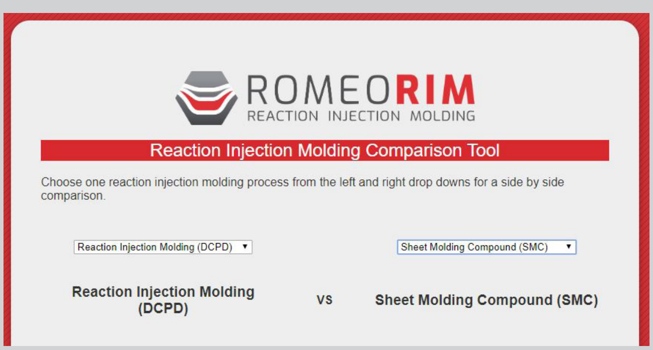
Romeo RIM Material Spectrum

Polyurethane	Flexible	Rigid
Structural Foam	Bayfit	Baydur
Solid Elastomer	Bayflex or Baytec	Prism, Baydur GS, Baypreg or Spectrim
Fiber Reinforced (LFI)	Bayflex, Baydur or Spectrim	
Dicyclopentadiene (DCPD)	(DCPD)	

Note: Romeo RIM offers a variety of other materials dependent upon customers' needs, including and beyond material supplied by Covestro, Metton, DOW Chemicals, UTC, Huntsman Polyurethane and BASF.

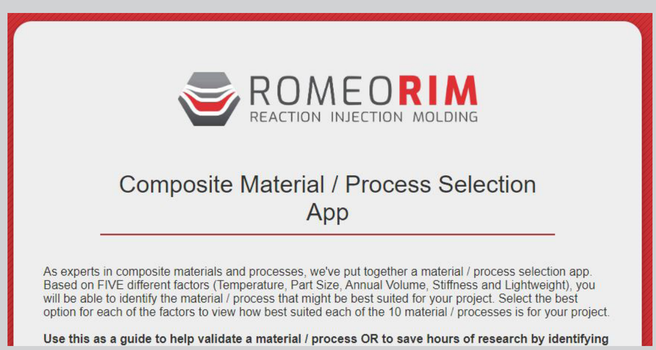
Material / Process Selection Tools

We've created two online tools to help you save time looking for the perfect RIM Material for your next project. To begin using it, simply enter the URL into your browser or click the image or link.



https://romeorim.com/RIM-comparison/

RIM Comparison Tool
Compare two RIM materials side-by-side



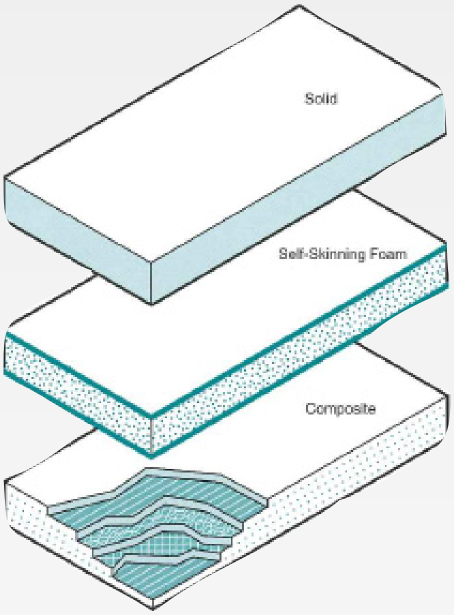
https://romeorim.com/selectionapp/

RIM Material Selection Tool
Enter up to 5 parameters and get a short list of suitable RIM materials

2.1 Material Selection

The level of rigidity typically defines a polyurethane system, which creates two categories: rigid or elastomeric. In general, a rigid polyurethane material has a higher flexural modulus and degree of hardness. This class of materials typically offers good thermal resistance, electrical properties, chemical resistance, and acoustical insulation values.

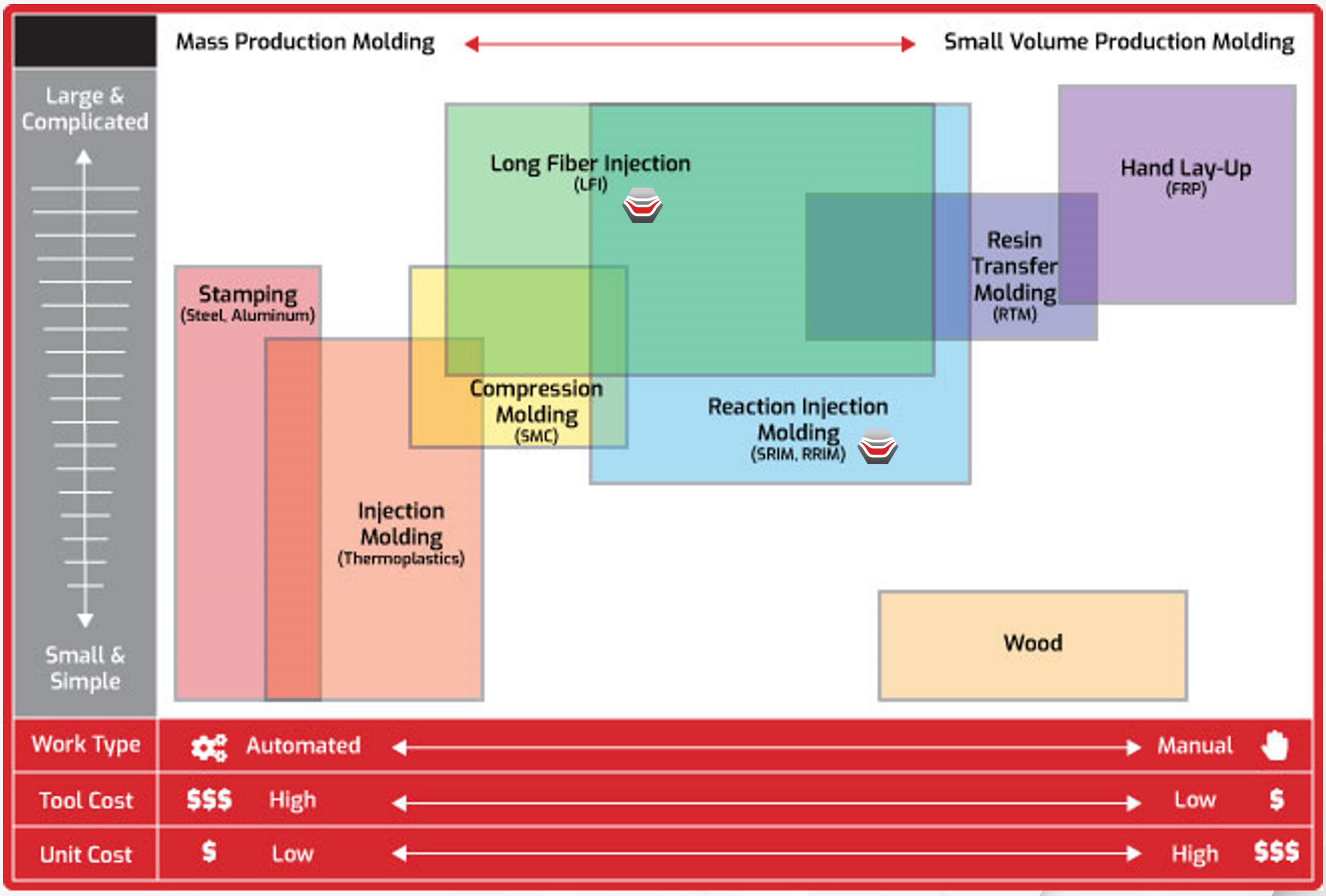
An elastomeric polyurethane system is often used in applications requiring superior impact strength. Elastomeric polyurethane systems display good toughness and dimensional stability throughout a wide temperature range and have excellent corrosion, abrasion, wear, and cut resistance. Physical properties for these two categories are not conclusive and the flexural moduli ranges of these materials overlap.



There are four types of polyurethane systems:

- Solid Elastomers
- Structural Foam
- Composite
- DCPD

A simple guide to help in material selection is listed below.

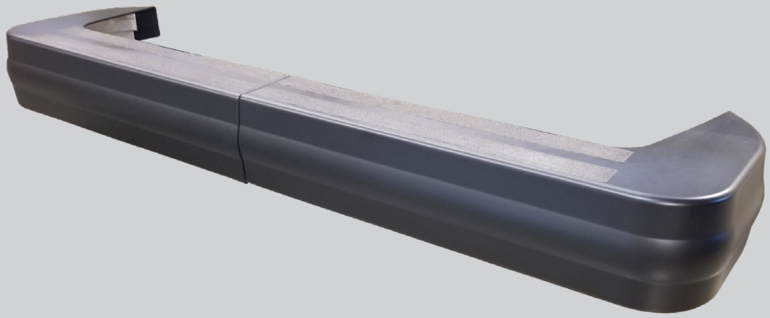


2.1.1 Solid Elastomer

Solid Elastomer systems do not use blowing agents (water, air, CO2 or Nitrogen), resulting in a homogeneous, rigid or elastomeric plastic. Solid elastomeric materials are used in many applications, including the automotive, specialty transportation, construction, agriculture, and recreational industries. Conventional parts include fenders, fascias, trims, and vertical panels – both interior and exterior. Glass or mineral fillers can be added to solid elastomers for improved stiffness. This is regularly referred to as Reinforced RIM or RRIM. These systems have been developed to handle higher-heat requirements - typically temperatures above +220°F.

Solid Polyurethane Characteristics

- Two-component, liquid systems
- Large part moldability
- Thick and thin wall substrates
- Stiffness
- In-mold coatings
- High-gloss finish
- Excellent surface quality
- Excellent chemical resistance
- Good Heat Deflection
- High Strength to weight ratio
- Design Flexibility
- Good Impact Strength
- Toughness/gravel resistance



Solid Elastomer Systems

Bayflex 110 Series

- Engineering Elastomers
- Short Gel Time for 30 Second Cure Time
- Current Applications: Bumpers, Fenders, Body Panels, Fascias

Bayflex XGT Series

- Engineering Elastomers
- Extended Gel Times
- Larger Parts
- Current Applications: Bumpers, Fenders, Body Panels, Air Dam, Fascias, Seat Base

Spectrim Solid Elastomers

- Good Impact Strength
- Design Flexibility
- Toughness/gravel resistance
- Current Applications: Body Panels, Fascias, Door Panels, Exterior and Interior Trim

Bayflex 180 Series

- High Flex Modulus
- Heat Resistant
- Fast Cycle Times
- Good Impact Strength
- Thin Wall Capable
- Current Applications: Cab Extenders, Panels, Chassis Fairings, Window Surrounds

Typical Properties of Bayflex Solid Elastomers

	110-25	110-50	110-80	XGT 100	180
Density (pcf)	65	65	65	65	65
Flexural Modulus (psi)	25,000	50,000	80,000	100,000	85,000
Tensile Strength (psi)	3,000	3,500	3,500	4,000	4,100
Elongation (%)	260	250	135	140	125
Notched Izod (ft-lb/in)	NA	11	5	7	5
Heat Sag (1 hour @ 250F, 4" overhand (mm))	NA	9	6	15	0.2
Doe C Tear (pli)	350	450	470	670	600
CLTE x 10E-6	110	110	110	110	85
Hardness (D scale)	50	58	65	69	69

2.1.2 Structural Foam

Structural Foam systems use a blowing agent to make parts with a higher-density skin and a lower-density, microcellular core in a sandwich-like composition. The blowing agent activates when the formulation's polyol component reacts with the isocyanate component. This forms a polyurethane with a cellular, foamed structure.

Structural foam and other rigid systems have hard, solid skins and are found in exterior panels, electronic and medical housings, automobile spoilers, skis, and other load-bearing applications. Structural foams are defined by their density. Polyurethane foams range in density from as little as 2 pounds per cubic foot to as much as 55 pounds per cubic foot.

The flexural modulus of structural polyurethane foams increase as the density increases. An unreinforced structural polyurethane foam with a density of 15 pounds per cubic foot contains a flexural modulus around 30,000psi. 40 pounds per cubic foot foam has a flexural modulus of 170,000psi. The highest modulus seen in an unreinforced structural foam can be 250,000psi.

Structural Foam Polyurethane Characteristics

- Two-component, MDI-based liquid systems
- Large part moldability
- Thick and thin wall substrates
- High Stiffness to weight ratio
- In-mold coatings
- High-gloss finish
- Excellent surface quality
- Styrene-free processing
- Lower material & Labor cost
- Higher productivity
- High impact properties
- Weight savings of 20-40% compared to traditional composites
- Low thermal expansion properties
- Mold behind Vinyl
- Soft Feel
- Sound Damping

Current Applications: *Engine Covers, Interior & Exterior Parts*

Typical Properties of Baydur Structural Foam Systems			
	671	**730**	**800**
Density (pcf)	45	40	61
Flexural Modulus (psi)	21,000	170.000	123,000
Tensile Strength (psi)	3,800	2,700	3,000
Elongation (%)	7	10	12
Notched Izod (ft-lb/in)	8	11	5.7
Heat Sag (1 hour @ 250F, 4" overhand (mm))	2.2	2.2	2.2
Doe C Tear (pli)	NA	NA	NA
CLTE x 10E-6	44	50	66.9
Hardness (D scale)	72	70	70

2.1.3 Composites

Composite systems are solid or foamed materials, molded in combinations with long-fiber reinforcements, such as glass mat or chopped, to improve the system's mechanical characteristics. Sometimes referred to as "SRIM", for Structural RIM, these systems have extremely high stiffness and high impact strength because of the filler. Typical applications include door panels, shelves, automotive load-bearing panels, and recreational equipment parts. At Romeo RIM, the process is a technology that injects the long glass fibers along with the polyurethane system as a one-step process, rather than the traditional two-step process of inserting preforms and mates into the mold. The glass is applied to the urethane at the mix head where it is chopped to the desired length and poured/blown into the mold. The mixhead, which is attached to a robot, is programmed to move over the open mold cavity while dispensing both the long glass fibers and the polyurethane. At the end of the pour, the mold is closed to form the part. This technology combines the materials used in RIM with the processing of SMC.

Long Fiberglass PU Composites

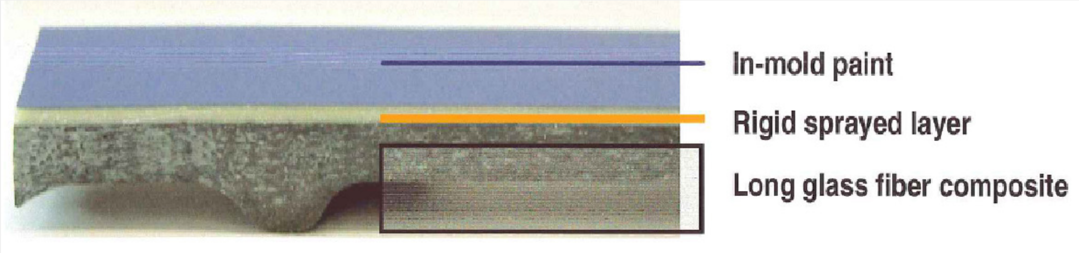
Long fiber Injection (LFI) is a spray molding based on the simultaneous pouring of polyurethane resin together with chopped fiberglass at different lengths.

Baydur & Spectrim Fiber Reinforced/Long Fiber Injection Characteristics

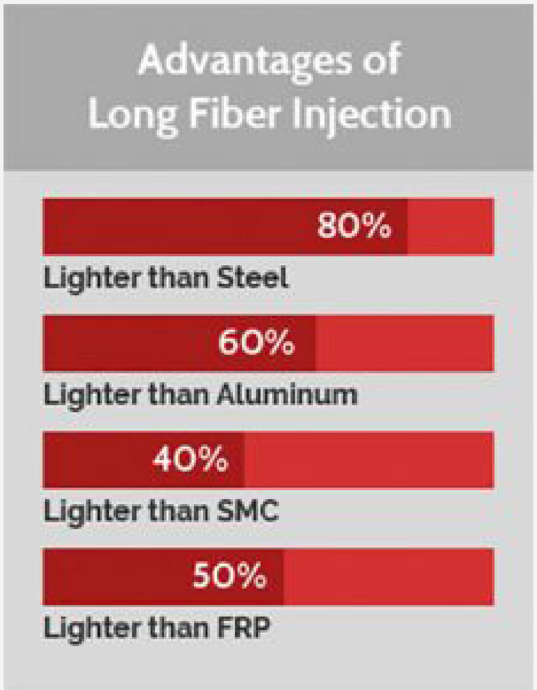
- Two-component, MDI-based liquid systems
- Large part moldability
- High strength to weight ratio
- Stiffness
- In-mold coatings
- High-gloss finish
- Excellent Adhesion
- Excellent Impact Strength
- Replaced the Traditional Fiberglass Mat by Mixing Chopped Fiberglass with Polyurethane
- Greater Versatility in Manufacturing
- Higher Physical Properties at Equivalent Weights
- Control of Glass Distribution
- Glass Adjustable During Pour Cycle
- Variable Part Thickness
- Use different fiber length
- Produce fit-finish components
- Parts produced with cycle time of 4 to 5 minutes

Current Applications: *Interior Cabinets, Door Skin, Instrument Panels, Pillars, Sleeper Cab Interior's, Agriculture Components, Aftermarket Components and Spa Cabinet Surrounds.*

The next generation of LFI substrates, High Gloss LFI, is providing improved surface properties when compared to traditional materials. This new technology is capable of reaching a Class A surface quality normally required by the automotive and transportation industries. These markets typically use steel, SMC and other thermoplastic film backed with PC, ABS and PVC. This technology incorporates a sprayed rigid layer that has good mechanical properties (high stiffness, high surface hardness and heat distortion resistance) with enhance surface qualities. This high density rigid layer is sprayed on the mold just before the LFI is applied to the mold surface. This rigid layer is capable of both in-mold and post applied painting processes. The picture above shows the layers of a high gloss substrate.

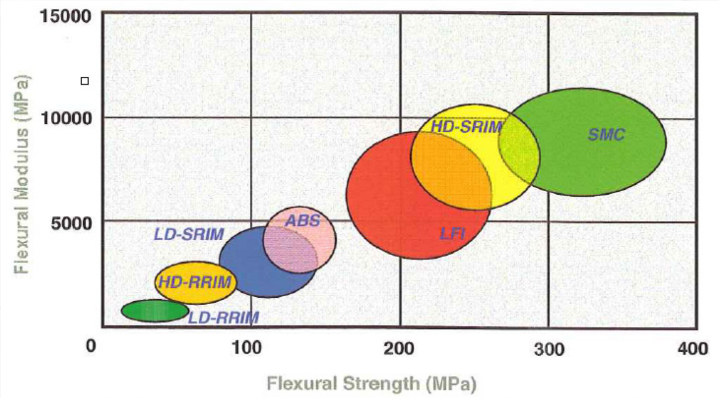


Typical Long FIber Injection Properties	
Weight (Specific Gravity)	1.12
Flexural Modulus (psi)	832,000
Tensile Strength (psi)	8,436
Elongation (%)	2.1
Notched Izod (ft-lb/in)	11.3
% Glass Content	39



Polyurethane has shown a good potential for growth in composites applications thanks to its fast production cycles, a wide range of mechanical properties and the possibility to be Volatile Organic Compound (VOC) emission free.

One challenge for further growth of polyurethane composites is the need to combine the fast and easy production processes with excellent surface finish. This allows for the final part to be painted directly into the mold or off-line; creating a competitive advantage over other thermoset composites such as Glass Reinforced Plastic (GRP), Sheet Molding Compound (SMC), etc. The chart at right shows typical mechanical properties of PU composites compared to other reinforced polymers.



Typical mechanical properties of PU composites and other reinforced polymers

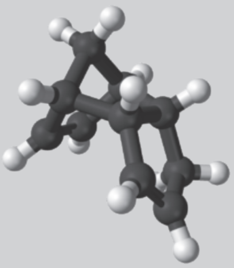
2.1.4 DCPD

The DCPD RIM process can provide large parts in low pressure molds with the mechanical property performance similar to injection molded engineered thermoplastics. The combination of DCPD process and material capabilities result in a new design freedom for replacing traditional materials such as metal, wood and fiberglass.

Injection molded engineering thermoplastics have a similar benefit package for smaller parts. Small parts for DCPD material are in the 8 to 10 ft^2 (1 m^2) range which is the upper end for cost and size capability for standard injection molding. In addition, DCPD parts do not require identical part geometry for multi-cavity molds.

DCPD Characteristics

- Surface areas up to 120 ft^2
- Large part moldability
- Variable wall thickness
- Part thickness up to 12 inches
- Low stress molding
- Fewer components vs. steel
- No rust or dents
- Excellent Impact Strength & Durability
- Outstanding surface finish
- Maintain tight tolerances
- Light weight and repairable
- High abuse applications
- Post applied painting needed for class-A finish.



The Mold Process

- Injection time is 15 to 20 seconds
- 700 pound (320 kg) Part Demonstrated
- Part Size is limited by machine flow rate and part thickness
- Part fully cured in 1 minute
- Cycle time is typically 4 to 6 minutes
- No post cure needed.

Current Applications: *Agricultural Components, Construction Equipment, Septic Tanks & Components, Heavy Truck Components & Fascias, Air Deflectors, Automotive Pallets & Racks and Medical and Electrical Housings.*

Typical DCPD Properties	
Density (g/cc)	1.03
Flexural Modulus (ksi)	273
Tensile Strength (ksi)	6.8
Yield Strain (%)	4.7
Notched Izod (ft-lb/in)	6
Plate Impact (ft-lbs)	34.1
Flexural Strength (ksi)	10.1
Tensile Modulus (psi)	275
CLTE x 10E-6	48.8

2.1.5 Fillers

Using materials that have glass and other inert fillers will affect your part's shrinkage, coefficient of linear thermal expansion (CLTE), stiffness, and impact strength. A filled elastomeric polyurethane material can have a CLTE closer to steel. Generally, fillers include fiberglass flakes, short glass fibers, or other mineral fillers. Usually, fillers need to have a sizing treatment to promote adhesion.

As filler content increases, stiffness increases as well as brittleness. Short fibers usually orient in the direction of flow, causing greater stiffness and lower CLTE parallel to the fiber orientation. Adding 15% glass filler to a elastomer can almost double its flexural modulus. Typical fillers used at Romeo RIM are Rimgloss, Wollastonite, Mica, Talc and Chopped Glass Rovings.

Selecting the Proper Material

When selecting a polyurethane material for your component, you must take into account the following items: Aesthetic considerations, Parts functional needs & Economic concerns. Note: Involving your Romeo RIM representative will aid in ensuring the correct material is selected.

Aesthetic Considerations

When establishing aesthetic requirements for your part, an important rule to remember is "form follows function". If your part is an exterior automotive component, your part will need a "class A" finish, because most automotive applications are aesthetically sensitive. In contrast, if you are designing an unexposed structural component, aesthetics may not be as important as the structural characteristics. While it is important to make your part look high-quality, specifying the finish or paints may increase your overall part costs. Some aesthetic guidelines to consider when designing parts include:

- Determine if your part will need a smooth, mirror-like, "class-A" finish, a textured finish, or other type of finish. This is important due to some polyurethane materials may be more suitable for your part than others.

- Determine if your part will require color. Polyurethane parts can be painted, pigmented, and in-mold coated.

- Determine if only pigmentation to the polyurethane material is acceptable. Darker colors –blacks, grays, or browns – may not need painting because color shifts caused by ultraviolet light will be less noticeable.

- Consider applying a good polyurethane paint or clear coat to prevent chalking or color shifts caused by UV exposure. Polyurethane coatings inherently have excellent adhesion.

Functional Considerations

When defining functional requirements, you must consider all the environments to which your part will be exposed to. Consideration must also be made for the end-use conditions. Ambient temperature, humidity, and UV radiation are of particular importance. Consider the following guidelines when addressing functional requirements:

- Determine the impact requirements: Does the part need high impact resistance and/or high stiffness?

- Consider elastomeric RIM polyurethane systems when your part needs good impact characteristics.

- Consider rigid polyurethane systems for parts that need high stiffness.

- Define the part's loading conditions, fastening or attaching parameters, and other physical requirements. The physical properties of the RIM material must withstand the structural conditions to which your part will be subjected.

- Determine the chemicals to which your part may be exposed during processing and assembling, as well as in end use. These include, but are not limited to solvents, degreasers, cleaning agents, and household products. Ensure that these chemicals are compatible with your material selection.

- Determine if stiffness inserts should be encapsulated in the part.

- Use a structural composite polyurethane if your part needs high stiffness and high impact strength.

- Use self-skinning, rigid, foamed systems when you need to reduce density and part weight.

- Consider the CLTE of the other components mating to or surrounding the part.

- Consider the heat requirements that the part will be exposed to.

Economic Considerations

The cost of the part you are developing will include more than the per-pound cost of the material you select. Different materials – steel and plastics – have different costs associated with processing, finishing, productivity, and quality control. All of these factors can alter your cost dramatically. RIM polyurethane materials can offer quick cycle times for large parts made of elastomers and, in several cases, can use less-expensive equipment than thermoplastics. Additionally, parts made of polyurethane may weigh less than comparable parts made of other materials. Polyurethanes have a great strength to mass ratio. Because the part's shape, not its weight, is fixed in the design, you should also compare the cost per volume ($/in^3) instead of cost per pound. A ton of low-density material will produce more parts than a ton of high-density material.

Part geometry also plays an important role. When comparing polyurethane systems for a load-bearing application, optimize part geometry for each material's characteristics. For example, you may be able to design a part with thinner walls and fewer ribs to achieve the required stiffness by using a higher-density system. Consider these guidelines when determining costs:

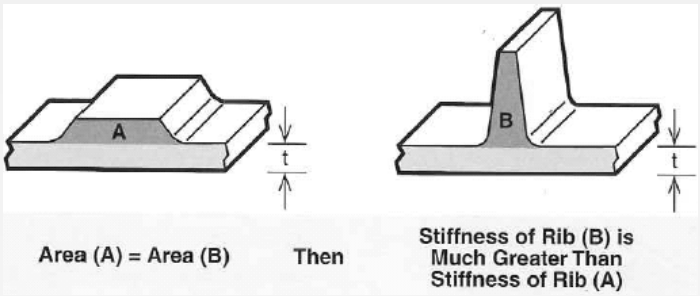
- Identify your cost target.
- Know the current cost of manufacturing and assembly, particularly when replacing a part made using a different process.
- Where possible, optimize all thickness to reduce part weight and mold-cycle time.
- Core thick sections where possible. Use other reinforcing techniques such as ribbing, corrugating, and encapsulating structural inserts to improve the part's stiffness.
- Simplify part design. Complex parts with multiple side pulls will increase mold costs significantly.
- Consider the added cost for finishing, painting, or coating.

2.2 Wall Thickness

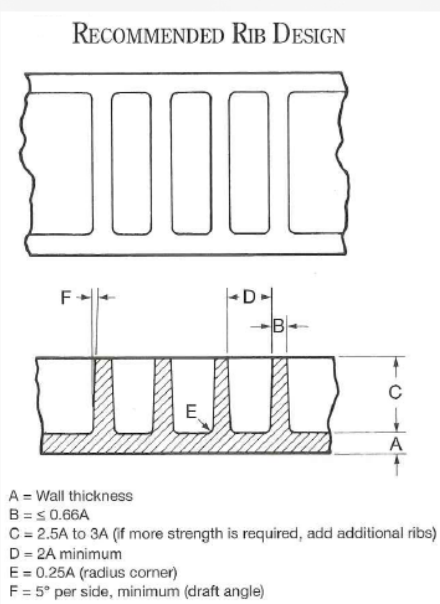
Polyurethane systems are better suited for parts with varying wall thicknesses than thermoplastics. Thicker wall sections allow for increased part stiffness. However, thicker wall sections drive increased curing times, longer molding times and increased material costs. Thin-walled parts have shorter cure times because the heat of reaction dissipates more rapidly. While RIM processes are flexible enough to make parts with varying wall thicknesses, designing parts with excessive wall-thickness variation may cause uneven filling and race tracking.

2.3 Rib Design

One alternative to thick wall sections is to add ribs. When designed correctly, ribs can stiffen a part without increasing part weight significantly. Taller, thinner ribs are more effective than shorter, wider ribs. Ribs should run continuously from side to side or corner to corner. Notching a rib or ending it mid part will reduce the effective stiffness of the rib.



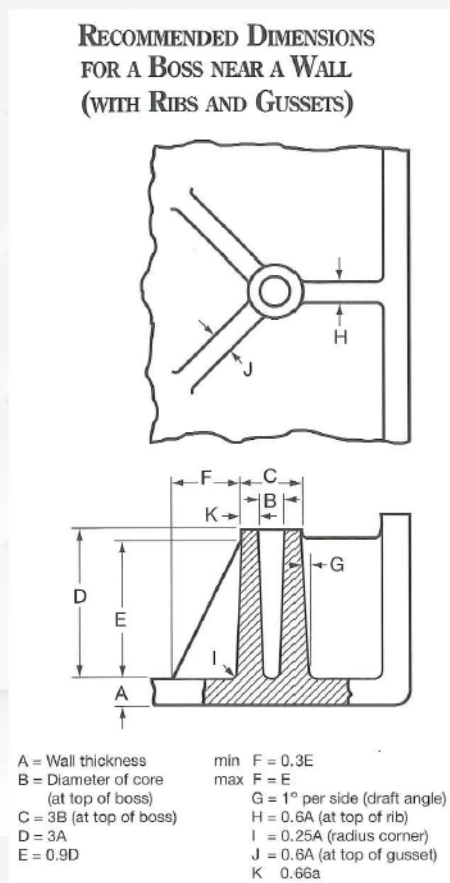
Taller ribs with draft may result in a wide rib base. This could cause processing issues, increased cycle time, and appearance issues (sink). Sink appears where the rib and mating wall meet because the increased wall thickness leads to increased local shrinking as the part cools. A maximum rib thickness of 0.66 times the nominal wall thickness should eliminate the likelihood of sink. Ribs that are greater than this can cause read through on the A surface. In general, sink marks and read through are less of a problem with RIM polyurethane than with thermoplastics. Designing a step in the part where the rib meets the mating wall helps to avoid sink marks. If you want the support of a thick rib; design it as a series of thinner ribs with the equivalent height and cross-sectional area. The space between the thinner ribs should be at least double the nominal wall thickness of the part. Whenever possible, place ribs behind style lines to minimize the effects of processing variables. It is recommended to draft each side of the rib wall 5°; however less draft is possible on ribs shorter than 25mm.



2.4 Bosses

When designing a boss it should be treated like a cylindrical rib and apply rib design guidelines to the wall cross section. Whenever possible attach bosses on the inside of parts to the side walls with connectors that allow air to escape during molding. Avoid isolated bosses whenever possible. If you cannot attach a boss to a side wall because of interference or distance from the wall, design gussets that extend from the base to the top on the side in the direction of flow or vent the boss with a core.

If using a boss for an insert, make the hole as deep as possible, preferably leaving only one nominal wall thickness to prevent sink marks. The recommended maximum height of a boss is three times the nominal wall thickness. Bosses should be at least two times the nominal wall thickness from any other features. A minimum draft of 1° should be used for the

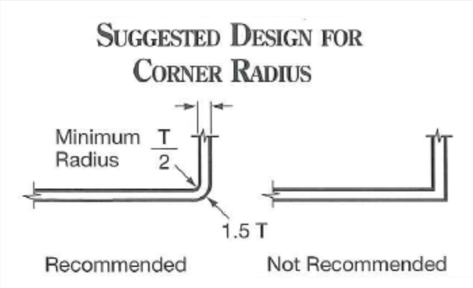


outer wall of the boss and all bosses should have radii at their bases.

Core bosses instead of drilling when using thread-cutting screws and thread-cutting inserts in parts made of structural foam and LFT to increase pullout strength.

2.5 Radii and Fillets

Maintain a uniform wall thickness through corners. In general, design the outside radius to be 1.5 times the nominal wall thickness, and the inside radius to be 0.5 times the nominal wall thickness. A minimum inside corner radius of 2.0mm is recommended.

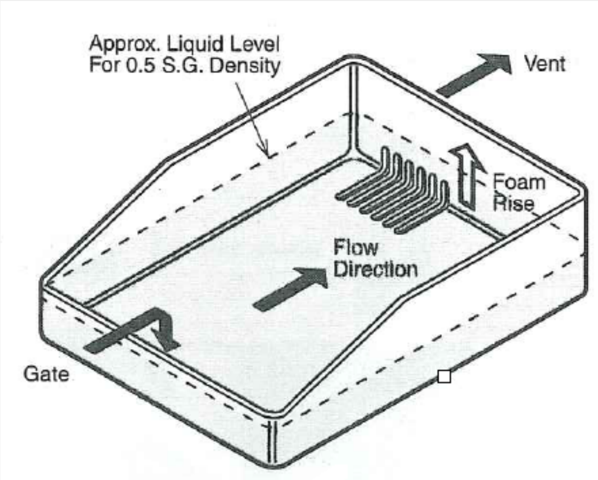


At intersections, where there is a mold parting line, sharp corners are possible; however, uniform paint coverage at any sharp outside corner will be difficult to obtain. For best results break sharp edges with a radius of 0.25 times the nominal wall thickness, or use a minimum radius of 1.5mm for painted edges.

2.6 Holes, Grooves and Slots

Holes can be formed in a variety of ways and in various locations. They can be post drilled, molded in die draw, or formed by a retractable pin actuated by a hydraulic cylinder. If the hole is in a side wall and has enough draft, then it could be formed by a split shutoff between the cavity and core of the tool.

Grooves and slots should be oriented in the flow direction of material whenever possible to minimize air entrapments or knit lines. Grooves should be rounded or chamfered rather than sharp to help flow, vent air, and reduce stress concentrations. Grooves should not reduce the wall thickness to the point it impedes the foam flow. Do not recess grooves more than 3/16" for foamed materials and 3/32" for solid materials. The width of grooves should not be

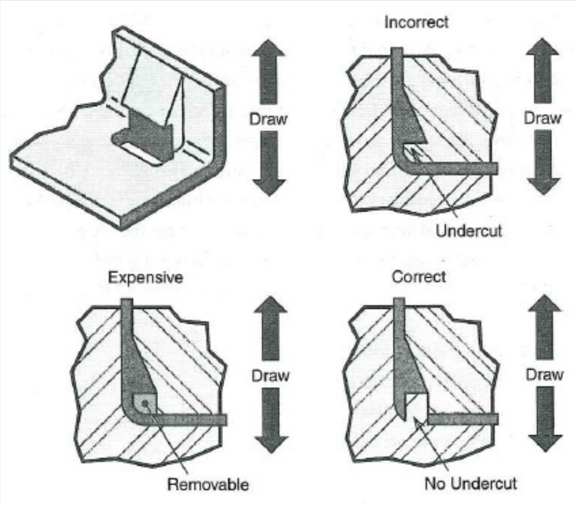


smaller than the nominal wall stock. However, wider grooves run the risk of race tracking and air entrapments. The width of material between parallel grooves should be 1.5 times the groove width. Groove length should not be more than 20 times the width of the groove.

Consider positioning slots in a side wall, curled around the base plane, to allow for molding without slides. Another option is to design slots with stepped cutouts, positioned in a sloping wall section. Thicker walls will require more slope. When using this option, do not make the mating sections too sharp, as this could damage the mold. Put a minimum of 1.5° draft on the groove walls to help with demolding. When designing grooves, slots, and holes in foamed materials, they should be located under the liquid level and lie in the direction of foam rise to help prevent air entrapment.

2.7 Undercuts

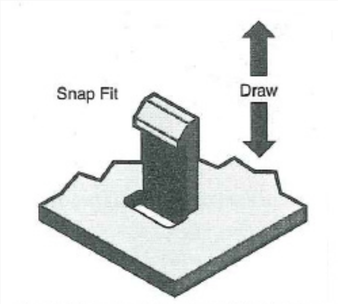
Whenever possible, avoid undercuts when designing parts made of rigid RIM polyurethane. They add to cost of tooling incorporating action and may create demolding problems. Modify the part geometry, mold orientation, or divide your part into two separate molds to avoid undercuts. For parts made of



elastomeric materials, minor undercuts can be a design advantage without affecting tooling costs. The flexible nature of these materials accommodates easy part removal even with the minor undercuts.

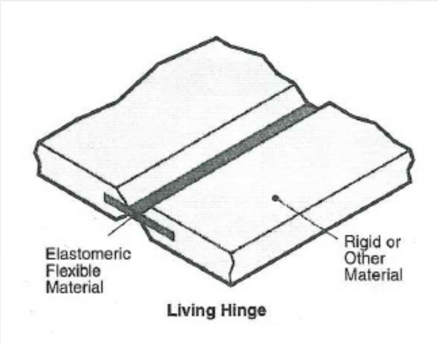
2.8 Snap Fits and Hinges

A simple, economical, and rapid joining method, snap-fit joints offer a wide range of design possibilities. All snap fits have a protruding part on one component - a hook, stud, or bead - which deflects briefly during joining and catches in a recess in the mating component, thus relieving the deflection force.



Snap-fit hook molded through hole to form undercut.

When designing hinges, consider the end use: Will it be a permanent connection? Will it be used often? Will it have to disengage after a certain opening angle? All of these factors will affect design. For permanent, frequently used joints, consider metal hinges, which can be molded-in or assembled after molding. While they add to costs, they may be optimum in long-term applications. For permanent, infrequently used hinges, consider the living hinge. Typically, they are made of the same material as the part, but can be made of different material. Elastomeric materials have excellent flexural strength. Molded strips of elastomers can be cut and placed into a mold to form a living hinge for a more rigid part. However, if such a hinge breaks, it will be virtually impossible to repair.



Living Hinge

Another-hinging method is to mold a part that looks and operates like a metal hinge, with alternating sections on opposite part halves. These partial hinges offer a designer a method of forming hinges without undercuts. While they have reduced load-carrying capability, partial hinges offer lower tooling costs and use hinge pins, as full metal hinges do. A rod pushed through the assembly completes the hinge. This design will disengage when the joint angle reaches 180 degrees. If you do not want the hinge to disengage, consider designing full holes at the ends and a retractable core pin.

2.9 Part Tolerances

Reaction injection molding materials fall into several classifications. This would include solid elastomers, structural foams, composite materials (long fiber Injection), and dicyclopentadiene (DCPD). The physical characteristics of these materials, with the exception of DCPD, can be manipulated with the addition of fillers such as wollastonite or milled glass fiber. The following dimensional tolerances would apply to all of these materials unless otherwise specified.

Specification of Tolerances on Part Drawings

The tolerance grade that applies to as-molded (unmachined) surfaces shall be specified in the general notes of the part drawing. Customary practice limits tolerances of reaction injection molded parts to those surfaces contacting the mold.

Dimensional Tolerances

Dimensional Tolerances are:

< 305 mm ± 0.5 mm

> 305 mm ± 0.002 mm/mm

Example:

Tolerance applied to a dimension of 700 mm:

Tolerance = ±(.002)(700 mm) = ± 1.4 mm

Wall Section Tolerance

The two most common wall section thicknesses are 3.2 mm and 6.35 mm. As wall thickness increases, there is increased shrinkage that leads to greater dimensional variation. For materials with a wall thickness of 3.2 mm, the tolerance that should be applied is ± 0.3 mm. For materials with a wall section of 6.35 mm, the tolerance that should be applied is ± 0.5 mm.

Flatness Tolerance

Flatness Tolerances are:

< 305 mm ± 0.015 mm

> 305 mm ± 0.0015 mm/mm

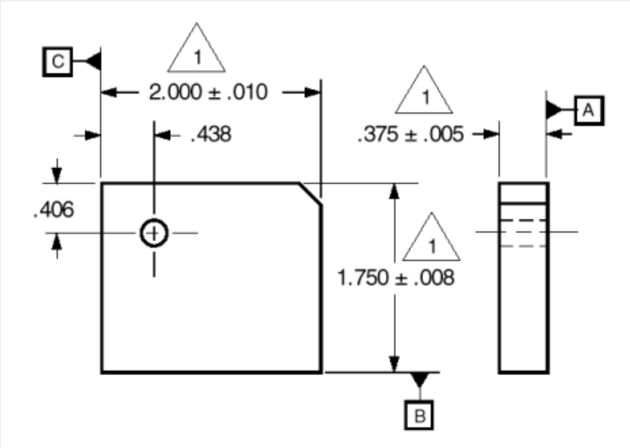
Example:

Flatness Tolerance applied to a part 700 mm long:

Tolerance = ±(.0015)(700 mm) = ± 1.05 mm

In-Mold Coating Tolerance

In-mold coating technology can be applied to all urethane systems with the exception of DCPD. Similar to a top-coating operation, too little paint and too much paint can have adverse affects on the surface finish of the part. The standard thickness of an in-mold coating in urethane systems is 2.5 mils with a tolerance of ± 1 mil.

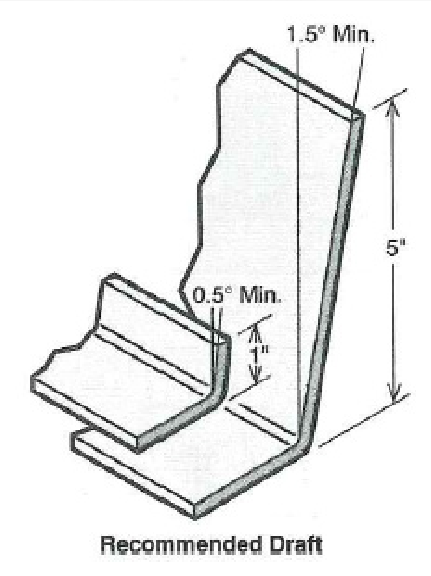


2.10 Draft

Every surface parallel to the direction of draw needs a draft angle to facilitate demolding. The recommended draft angle increases with part height. It is also recommended that the tool have a good polished surface to improve demolding.

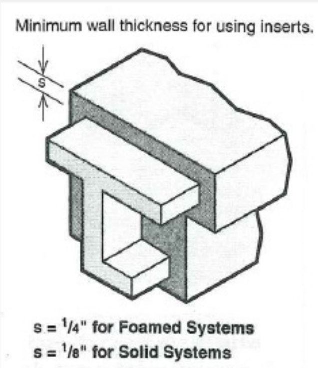
Generally, draft is more important on the core side (usually the top half of the mold) than it is on the cavity side, because parts generally shrink onto the core during cooling. Other rules of thumb for draft angles include:

- A minimum of 1/2 degree is usually adequate for parts with low side walls or ribs, typically those up to 1" deep.
- Add at least 1/4 degree of draft for every additional inch of draw, such that a 5 inch draw would require a minimum of 1-1/2 degree draft.



2.11 Insert Molding and Encapsulation

Polyurethanes have low molding temperatures and pressures, making them ideal for encapsulating reinforcing inserts. The insert should not impede material flow. If using a hollow insert, the ends must be sealed.



Thermoplastic end caps have been successfully used to seal inserts. To promote good adhesion with the polyurethane, clean and roughen the inserts and treat them with an adhesion promoter, if necessary.

The type of RIM system used determines the recommended minimum distance between an insert and the mold wall. For solid materials, this distance is 1/8 inch. For foamed systems it is ¼ inch.

Encapsulated inserts have the following benefits:

- Increased stiffness
- Reduce wall thickness
- Absorb high stresses
- Control Thermal Expansion
- Protecting Electronics

Metal Stiffening Inserts

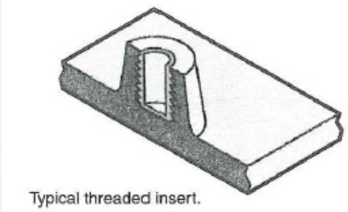
Molding metal inserts into RIM polyurethane materials will increase stiffness significantly. Inserts of all types (flat plates, extrusions, tubes, and bars) can be encapsulated. Fully encapsulating inserts eliminates metal corrosion, while reducing thick cross sections, controlling deflection and thermal elongation, and absorbing high stresses. The center of gravity for the insert should coincide with the center of gravity for the RIM material to reduce the potential for warping. This is due to the differences in the coefficients of linear thermal expansion in the materials. As the temperature increases, the polyurethane material will be in compression and the metal insert will be in tension. As the temperature decreases from ambient, the opposite happens. The relative cross-sectional areas of the two materials determine the ultimate elongation of the part.

Wood Stiffening Inserts

Typically cheaper than metal inserts, wood inserts can also be used to stiffen polyurethane parts. When a finished part is subjected to repeated loads, wood inserts may separate from molded polyurethane if the wood's moisture content is higher than 6%. In cases where the wood cannot be dried to reduce the moisture, it must be sealed with a lacquer before molding.

Threaded Inserts

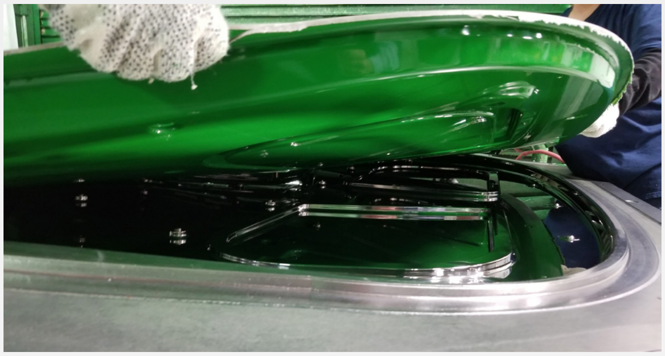
Threaded inserts are particularly useful when components must be attached to RIM parts. Use appropriately sized, press-fit inserts with respect to boss-hole diameter. Use threaded inserts if your part is going to be frequently assembled and disassembled. With structural foam, molded-in inserts my offer greater pullout strength, because skin forms over the entire insert surface. When using press-fit inserts with structural foam parts, mold the hole so that skin forms inside.


Typical threaded insert.

Generally, press-fit inserts are preferred to molded-in ones, even though they are not as strong. Placing inserts on pins inside the mold can increase cycle time significantly. Inserts have also been known to fall off pins during molding on rare occasions. The risk of potential tool damage must be weighed against the need for additional strength. The insert design, hole diameter, part density, and thread size determine the pullout force and stripping torque of threaded inserts.

2.12 In-Mold Coating

Consider in-mold coatings – special paints sprayed onto the mold surfaces –as an alternative to postmold painting. After spraying, these paints dry for a brief period,



so that the injected mixture flows over the semi-dry coating during mold filling. Typically in-mold painting is used for large, relatively simple molds, such as cab roofs and fenders for agricultural combines. Other points to consider when selecting an in-mold coating include:

Advantages of In-mold Painting

- Almost perfect adhesion between the coating and the substrate.
- Because in-mold systems are low in solids they follow the contours of the mold design very well. Therefore, very fine details on the mold will be faithfully reproduced.
- Capable of flexing with the part with no results of cracking and crazing in the part surface.
- Mold spraying is relatively straight-forward and less labor intensive than post-mold painting.
- Paint usage is relatively low compared with post-mold painting.

Disadvantages of In-mold Painting

- Complex mold shapes are difficult to spray with acceptable accuracy or speed.
- Unlike post-mold painting where a specific area of the factory can be set aside for spray finishing, in -mold painting is generally undertaken on the molding line. However, Romeo RIM's manufacturing clamps are set-up with all necessary equipment for in-mold painting.
- If the mass color of the polyurethane system is not a particularly good match for the paint finish, the flash or split line will stand out and may require touching up with a post-mold paint. (ie – black substrate with red topcoat)

2.13 Textures

Polyurethane molding techniques accommodate a number of different textures. Tools made of Aluminum, P-20 or nickel-shell are capable of having texture added to the surface. Texture accomplishes visual effect, improved moldability and has functional characteristics.

Visual Effect

- To give the part the appearance of leather, wood, stipple, sand, or whatever material you are simulating.
- To give the part a more even, planned effect, or to get rid of a glossy appearance and change to a matte finish. This can add richness to a part's appearance, therefore making the part more marketable, and giving it a perception of higher value and quality.
- To build into the appearance of the part a company's logo or a pattern that immediately identifies the part as belonging to that particular corporation.
- For visual contrast - through the use of two different textures on one part or by frosting the background or foreground of a logo, for example.
- Texture can provide visual improvement of a tough to mold part. Certain

textures can hide flow lines, knit lines, blush marks, and other molding flaws. Sometimes, even sink marks can be disguised by the application of texture.
- Texture can add a logo, a part number, a design, instructions for consumers, or part identification, thus eliminating secondary processes such as hot stamping or application of labels.

Improving Moldability

- Adding texture to a core can help to hold the part onto the core without manual undercuts which could create sink marks. The texture disperses the pressure over a larger area, lessening the likelihood of sink marks and yet still holding the part, allowing the mold to eject properly Thus, lessening the potential for drag or scuff marks on the Class "A" surface of the molded part.
- Texture applied on the core side and across lifters and/or slides, for some materials, can hide the shadow marks which sometimes will show through from the front of the part when the Class "A" side is polished.
- Texture applied to some molds will allow trapped gasses to escape more quickly, by venting to the parting lines from within the cavity.
- Texture is sometimes applied in order to better hold paint during a secondary molding operation.
- Texture applied in the correct design and location can help to minimize turbulence created by urethane flow.
- Texture can provide a functional rough surface finish, on a roll for example, to help the roll stock through the rolling process.

Functional Characteristics

- Texture adds strength to parts.
- Texture adds thickness to the parts - Typically from .030" to .100" of metal remove evenly over the surface of a mold component.
- Non-slip textures can be applied to add safety measures to a part that requires that particular quality.
- Texturing a fine texture on label areas of a part can help the labels to stick to the finished part.
- Texture creates more surface area. This can be useful in a fan blade (for example), since you don't change the size of the part, but you do push more air with the same blade. This principle could also apply in situations where more surface area will draw heat or cold away from the surface.
- Some textures just feel good to the touch or provide "grip," such as on a handle of a power tool, or handles for ski poles, etc.

2.14 Letters and Logos

Graphics can be molded into a part during manufacturing. Typically these visual elements are milled into the mold resulting in a raised appearance on the finished part.

Simple masking will allow raised graphics and lettering to be painted in a contrasting color. For dense areas of text or small letters, consider using decals. These decals can be placed into the mold and bonded to the substrate upon the injection of the material.

2.15 Shrink Rate

Mold shrinkage, listed as length-per-unit-length values or as percentages, assumes room-temperature measurements. Many processing and design factors determine the amount of shrinkage for a given application. Use published shrinkage information with caution as it is tested under laboratory conditions that may not reflect your specific part geometry or processing environment. Consider the following when addressing shrinkage:

- Cooling rate and mold temperature can affect the level of shrinkage in resins.
- Thick-wall sections cool more slowly

and tend to shrink more than thin-wall sections;
- Fiber-filled materials typically exhibit much less shrinkage in the flow direction;
- Mixed orientation typically leads to shrinkage ranging between published flow and cross-flow values.
- Shrinkage varies with the level of packing.

Packing forces additional material into the mold to compensate for volume reduction, lowering shrinkage. Packing typically decreases and shrinkage increases further from the gate, particularly in distant thick-wall sections. The mold constrains the part and prevents significant dimensional change until after part is removed from the mold. The type and duration of this constraint can affect net shrinkage between part features. Long cycle times constrain the part in the mold longer and reduce initial shrinkage, but can induce stresses that lead to additional shrinkage over time as the stresses relax-post mold cure.

Many factors can affect the level of shrinkage. You can usually obtain the most accurate shrinkage values for new molds by calculating the actual shrinkage in existing molds producing similar parts sampled in the same material.

Published shrinkage data represents the typical range of shrinkage based on laboratory conditions. Applying this data to a specific part and mold requires a combination of engineering judgment and estimates based on best practices. Tend toward the lower end of the range for parts thinner than 0.100 inch, and for highly constrained features such as the distance between holes. Anticipate flow orientation in glass-filled parts and apply the flow and cross-flow shrinkage values appropriately. Areas of random orientation will tend to shrink at a level midway between the flow and cross-flow values. Consider designing critical features and dimensions "steel safe" to simplify modifications to correct for errors in shrinkage prediction.

Geometric asymmetry (e.g., a flat plate with a large number of ribs that are aligned in one direction or on one side of the part) will introduce non-uniform cooling and differential shrinkage that can lead to part warpage.

Unfilled Materials: On the other hand, if an unfilled molded part contains high levels of molecular orientation, shrinkage is anisotropic because aligned chains shrink to a greater extent in the direction of orientation.

Filled Materials: For fiber-filled components, reinforcing fibers inhibit shrinkage due to their smaller thermal contraction and higher modulus. Therefore, fiber-filled materials shrink less along the direction in which fibers align (typically the flow direction) compared to the shrinkage in the transverse direction. Similarly, particle-filled parts shrink much less than unfilled grades.


INJECTION CONTROL
OPERATION
MANUAL
BALL VALVES
AUTO
A CYLINDER
B CYLINDER
INJECTION READY
CYCLE INTERRUPT
SETUP KEY
CLAMP #1 INJECT
STOP INJECTION


NOTHING

Chapter 3 - Tool Design

3.1 Tool Construction

When designing molds for use with RIM polyurethane systems, you must address several issues, including mold size and cost, clamping pressure, part shrinkage, dimensional tolerances, and part repeatability.

When designing molds, try to weigh the mold cost against the production volume and cost of post-molding labor needed to finish the part. Post-molding operations, such as trimming, drilling, bonding, sanding, and painting, can add significant cost to a part. Designing a more complex mold may reduce overall cycle time and post-molding labor. While the mold may cost more initially, it could save money over the production life of the part. Thus justifying the higher initial expense. Less complex parts that are flat and have minimum draw will have the lowest tooling costs.

3.2 Mold Material

Because RIM polyurethanes generate heat when they react, you should choose a mold material that is conductive. This is required to dissipate heat from the curing part. Aluminum and steel P-20 molds are ideal for production. With Epoxy and spray-metal molds being poor in heat dissipation and in poor surface quality, they are recommended for only prototyping and low-production runs. For high volume production runs, particularly those using reinforcing fillers, steel and aluminum are mold materials recommended.

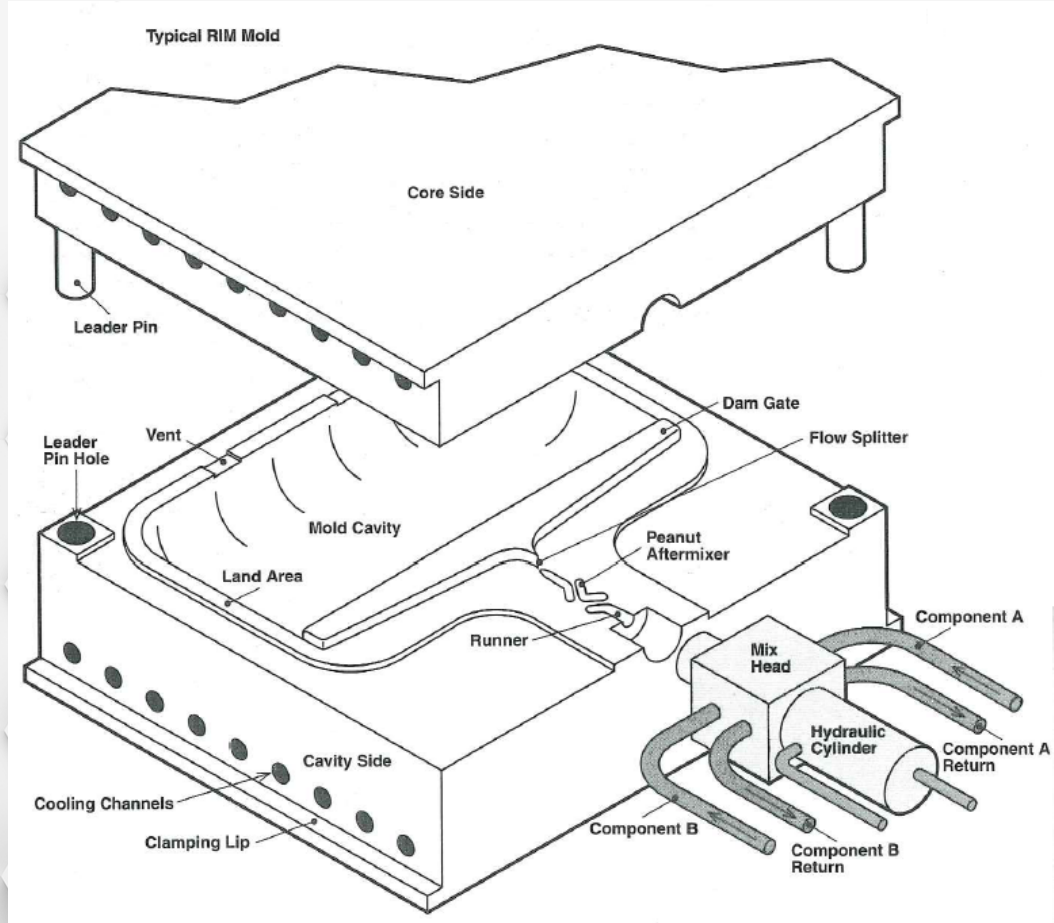


Chart below shows comparison of different materials and fabrication techniques for molds:

The RIM process allows you to use softer mold materials. Therefore, part features can be machined directly into the mold. This minimizes all four cost factors by avoiding the added costs of EDM machining, inserting, benching, polishing and hardening. Minimizing mold cost also minimizes the lead time. The complexity and construction materials determine the majority of the mold-making cost and consequently a large share of the finished-part cost.

3.3 Tooling Considerations

As part of the critical path of designing, engineering and prototyping a new RIM panel, tooling becomes a critical issue. The following questions must be answered before a tool can be properly developed:

1. What are appropriate tool considerations for manufacture of the parts?
    - Life of tool
    - Number of parts to be molded
    - Surface finish requirements (high gloss or low gloss paint)
    - Single or multiple cavity molds
    - Mold cycle time and heat conduction
    - Part tolerance and dimensions
    - Size and complexity of parts
    - Timing to completion of tool
    - Overall quality of parts to be made

2. What is the design of the aftermixer?
    - Harp or peanut

3. What is the gate size and location?

4. What is the size of the mixhead? What is the make of the mixhead?

5. Where are the locations of the parting lines, vents?

6. Where are the areas of critical tolerance? – Parallelism, Flatness and Style Lines.
Upon tool kick-off, document which dimensions are critical and which have looser tolerances. Prioritize them from most critical to nominal. Specified dimensions can influence tooling quotes.

7. What is the shrinkage of the polymer formulation? On the tool and after post curing.

8. What is the nominal part thickness?

9. What is the part weight?

10. What are the molding temperatures and pressure?

11. What fixtures are required? –Holding/Cooling, Degating/Trimming or Post curing?

12. What are the surface finish requirements of the core and cavity?

13. What is the cooling line pattern, including the slides and mixhead block?

3.4 Gating

After the material flows through the after mixer in the runner system it enters the gate. The gate transitions the flow of material from the cylindrical form to a planar form which can then enter the mold cavity parallel to and along the part surface. It is very important that the material exiting the gate is in a laminar flow condition. This is necessary to avoid spraying material into the cavity that will trap air and possibly cause defective parts.

The two types of gates typically used are Dam Gates and Fan Gates. The dam gate equalizes the material flow over the length of the gate prior to any material entering the mold cavity. This is accomplished by means of a well, a dam, and a splitting nose. The well is the area behind the dam. The cross sectional area of the well is approximately the same as the full incoming runner system or slightly larger. This is important to reduce backpressure, which can decrease mixing performance in the mixing head. The splitting nose or flow splitter is incorporated into the leading edge of the dam. Its purpose is to divide and direct the flow of material to either side of the well. The splitting nose serves to reduce turbulence in the material flow as it enters the well. Turbulence can result in entrained air in the material near the split. The entrapped air will be distributed throughout the part resulting in significant surface defects.

The Fan Gate gradually flattens the cylindrical form of the runner system to a planar form and bends onto the direction of the mold wall. Fan gates can be triangular or quadratic in shape. The apex angle of the triangular fan gate should not exceed 40 degrees. Beyond this limit, the stream entering the fan may separate from the sidewalls of the fan causing turbulence and generating bubbles, which show up as a defect in the part.

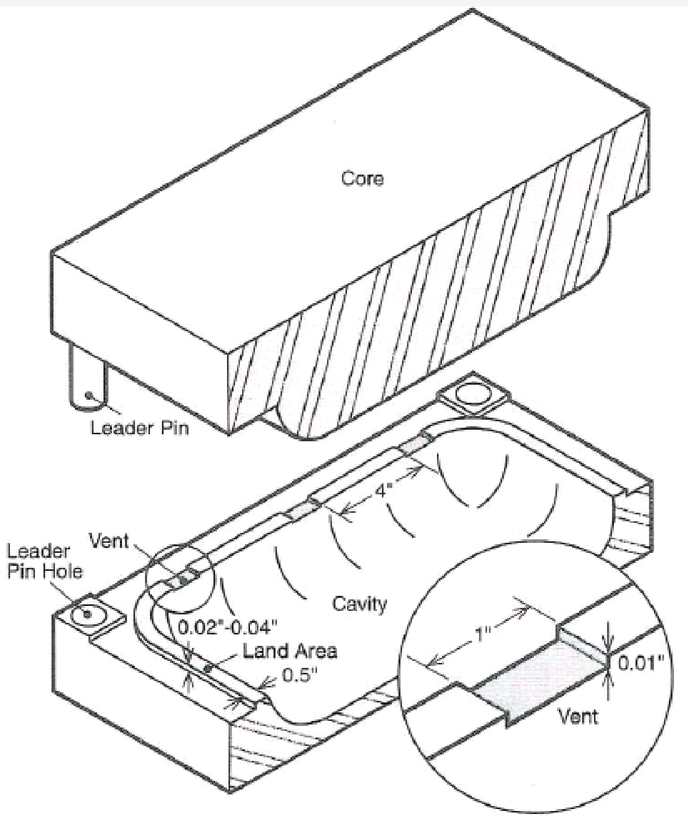
A quadratic fan gate provides a continuous transition from the full round shape of the runner system to the planar form of the gate with a constantly increasing cross sectional area. Determining what type and where to place a properly designed gate for a RIM project is the most important aspect of the whole part and mold design process. Gate placement can make or break a successful mold startup. The following items must be addressed in determining the optimum gate placement.

1. The type of polyurethane RIM material being used for the project.
2. The thickness of the part at the parting line.
3. Louvers, windows or other obstructions to material flow.
4. Ribs or channels that may dictate the direction of the material flow.
5. Parting line considerations such as stepped or rounded surfaces that would limit gate placement options.
6. The lowest possible point in the mold.
7. The shortest possible material flow length.
8. Part aesthetic considerations such as tolerable de-gating scars.
9. Multiple cavity molds.
10. Use of in-mold paint, primer.

3.5 Flow and Venting

The lowest point in the mold has always been the ideal location for the gate. When using a polyurethane structural foam part, it is a standard rule to gate into the lowest point in the tool. This is a necessary practice due to the final filling of the mold cavity to the desired part density is done with the pressure generated by the polyurethane system as it foams.

All molds used in manufacturing RIM polyurethane materials must have vents at the highest point in the mold cavity. This is required to allow the air in the cavity to escape during the injection of the material during the molding process. A common rule in developing vents is to make the vents wide and shallow rather than narrow and deep.



Typical mold venting using 4-inch spacing. Vents on high side of mold.

3.6 Typical Mold tolerance:

Molded Dimension:

For basic part geometry:
+/- 0.001in./in. or +/- 0.005in., whichever is greater

For complex geometry:
+/- 0.002in./in. or +/- 0.005in., whichever is greater

Flatness:

Restrained: +/-0.0010 in./in.
Unstrained: +/- 0.0015 in./in.

Post Machine Dimensions: +/- 0.005in.

3.7 Lifters, Slides, and Core Pulls

Hydraulically controlled core functions aid in part removal of tight-angled flanges and flares in the mold. It is best, however, to keep core functions simple and few in numbers. Parting lines in the core function sections may read through to the show surface. When properly designed, core functions can allow both easy access to and removal of a large complex part.

3.8 Critical Tolerance Areas

Each tool being designed has certain areas that need to match other parts in the assembly, or have critical aesthetic properties. For example, a flat part may require a polymer that shrinks the same in all directions to avoid distortion of the panel. This means mica will be the required filler. Often, style lines will be matched along the complete side of a truck, car or tractor. Front fenders may require a constant gap to the mating component. Material suppliers have computer modeling capabilities that assist in meeting these close tolerances.

Because of the high modulus and shrinkage on the tool, body panel tools require moving slides and cores to allow their release from the tool. The final part size occurs after the postcuring operation, thus becoming stable.

3.9 Multiple Cavity Molds

RIM molds can be designed with multiple cavities to manufacture several parts simultaneously. Multiple cavity RIM molds typically do not have more than four cavities – Two cavity molds are more common in industry. Typically, multiple cavity molds are cost effective for larger production runs. While the tooling may be more expensive than single cavity molds, production time and costs can be lower with multiple cavity molds.

Chapter 4 - Post Molding Operations

4.1 Trim Fixtures

Fixtures are required to support the part from processing to assembly and even during assembly in some instances. These fixtures will vary depending on the performance criteria. At the mold, the fixture needs to keep the part from distorting, not scratch the part surface, and leave open areas for trimming or sanding.

4.2 Postcure Fixtures

These fixtures should be the correct shape and size of the actual part. Oversized postcure fixtures will distort the part through the postcure operation.

4.3 Paint and Prime Fixtures

The prime and paint process requires fixtures that will hold the part in position and provide adequate grounding.

4.4 Post-Applied Painting

While more costly than pigmentation, post-mold painting offers the added benefits of exact color matching to other parts and parts

made of other materials. Postmold painting covers minor surface blemishes – dependent on the amount of solids in the paint formulation.

Advantages of Post-mold Painting

- Paint finishes with excellent exterior durability and chemical resistance can be produced. Achieved through additional topcoats.

Disadvantages of Post-mold Painting

- Poor adhesion between the coating and substrate is the most common failure area for post-mold paints. In order to avoid such failure extensive preparation of the substrate is necessary before painting. Such preparation includes removal of all traces of release agents by solvent washing, followed by application of one or more primer coats.
- The application of post-mold paints is a skilled operation. Faults in the top coat are all too easily produced (i.e. craters, orange peel, etc.). Operations to correct such faults can be costly in both time and labor. The standard of cleanliness in the spraying area needs to be high in order to avoid contamination by foreign matter on the substrate or wet paint surface.
- Since most two-component systems use isocyanate prepolymers as a curative, the health risk is somewhat higher than that associated with fully-reacted systems.

4.5 Assembly

Many customers rely on Romeo RIM to assemble components or entire products that utilize the parts we have manufactured. Assembly processes include adhesive bonding, riveting, electrical wiring and many types of mechanical fastening methods. Romeo RIM's assembly operations are a convenient and cost-effective way for customers to save time and money by not having to accommodate every manufacturing and assembly operation in their own facilities. The following items need to be taken into account when designing new products:

1. Simplify the design and reduce the number of parts.
2. Standardize and use common parts and materials.
3. Design for ease of fabrication.
4. Design within process capabilities and avoid unneeded surface finish requirements.
5. Mistake-proof product design and assembly (poka- yoke).
6. Design for parts orientation and handling
7. Design for ease of assembly
8. Design for efficient joining and fastening.
9. Design modular products.
10. Design for automated production.
11. Incorporate as much assembly into the mold as possible- Insert Molding.

4.6 Bonding

Polyurethanes or epoxy adhesives work well with RIM polyurethane systems. The adhesion area of the joint should be at least three times the wall thickness. Bonds can have high strength in both tension and bending. Clean and roughen the adhesion areas to promote good bonding.

Advantages:

- Moderately priced
- Excellent toughness and flexibility
- Good flexibility at low temperatures
- Excellent adhesives for a wide range of polyurethanes
- One/two component, room temperature or heat-cure adhesives are available.
- Varying cure times.

Limitations:

- Some bonding materials have poor temperature resistance
- Sensitive to moisture both in cured and uncured state
- Two component mixing or single component toxicity
- Short pot life
- Require special equipment to mix and dispense.

Adhesive bonding: Adhesives have been used to assemble composite components, such as inserts to assemblies and secondary brackets, and are sometimes used to join structural components. Bond reliability of adhesive joints is sometimes questioned for some composite applications. Three adhesives are often used to bond composites: epoxies, acrylics, and urethanes. Epoxies are especially reliable when used with epoxy-based composites because they have similar flow characteristics.

Careful preparation of the adhesion surfaces is essential to making a quality adhesive bond, but it varies depending on the adhesion and adhesive used. Recommended preparation of composite adhesion consists of a solvent wipe, to remove loose surface dirt and oil, and an abrading operation. Abrasion should be done carefully to avoid damaging composite surface fibers.

In some cases, primer is required to coat the composite before applying the adhesive. When bonding composites to metals, the metal substrate can be prepared by blasting with sand, grit, or metal oxides; abrading with a wire brush; and machining or scoring with cutting tools. Metal surfaces can also be prepared chemically. To protect freshly prepared metal surfaces from corrosion and contamination, adhesive should be applied as soon as possible.

4.7 Fasteners

One of the most commonly used methods of attachments in the polyurethane field is screws. Screws can be installed to a component by either through holes, molded-in bosses and self-tapping into the substrate. All three methods of attachments yield relatively high pull-out strength in the substrates. In most cases, the pull-out strength is proportional to the screw depth. When ever possible, the best practice is to mold-in a pilot hole versus post drilling a hole. A molded-in pilot hole will yield higher pullout results. Regarding pull-out strength, each material will very due to the overall thickness, filler type and percentage. Note, if the component will experience a high frequency of disassembly, consider another joining method i.e.– threaded inserts.

ROMEO**RIM**
REACTION INJECTION MOLDING

We don't just provide BIGGER, STRONGER, LIGHTER solutions.

We help make the products that make a difference in the world.

1990	1993	1996	2000	2009	2015	2017	Beyond
Innovated In Mold Painted (IMP) RIM	1st DCPD Molder Heavy Truck Marktet	Innovated Closed Mold HIgh Gloss IMP	Innovated Long Fiber Injection (LFI) 1st in N.A.	Innovated LFI High Gloss IMP	Innovated Low Cost Tooling	Innovated 2-tone IMP & Surface Finish	Coming Soon: **Project X**

ROMEO RIM

REACTION INJECTION MOLDING

74000 Van Dyke, Romeo, MI 48065 · 586-336-5800 · romeorim.com