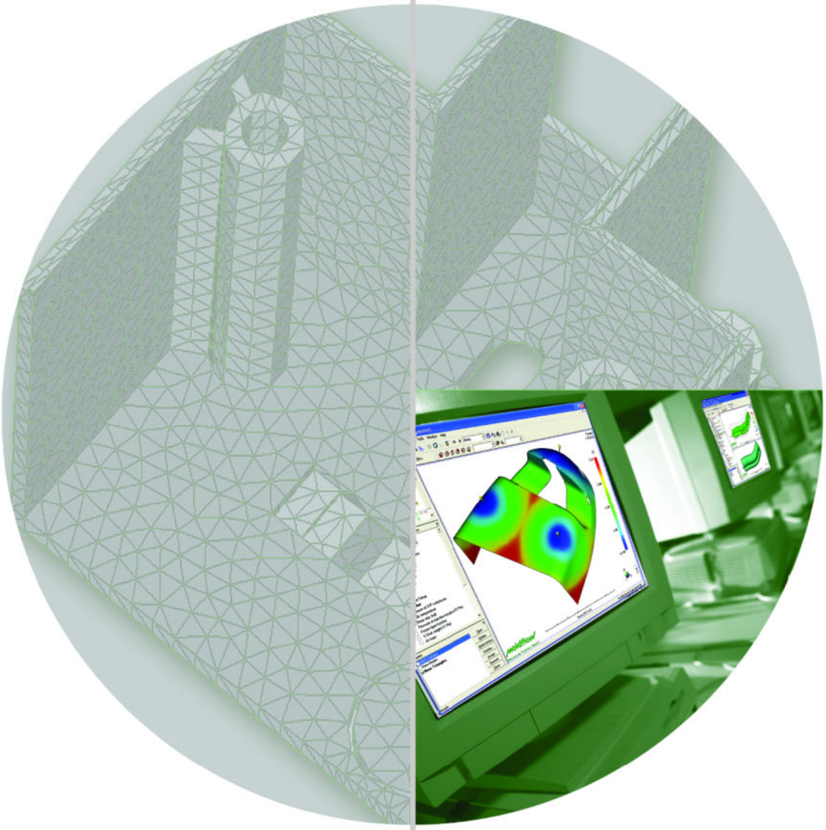
Moldflow Plastics Insight®

Release 6.0



Simulation Fundamentals Training
Theory & Concepts

Simulation Fundamentals

THEORY AND CONCEPTS FOR MPI 6.0

March 2006

Published by Moldflow Corporation,

Contents

About this manual

The Simulations Fundamentals, Theory and Concepts manual is designed with the new Moldflow user in mind. In creating this manual, our goal was to introduce you to some basic plastic flow and design principles and concepts related to translate, analyze and interpret models.

There is a significant amount of information in this manual, more information than can be absorbed during the class. This manual should be useful as a handy desk reference when back in the office.

Using this manual

This manual is separated into several chapters and appendices. Each of the chapters covers a specific topic and includes the following sections:

Aim

Describes the learning objectives of the chapter.

Why Do It

Outlines the reasons for following the prescribed guidance, suggestions, and methodology within the chapter.

Overview

A complete outline of what will be covered within the chapter.

Theory and concepts

Theory and concepts discusses background information on the subject matter of the chapter. This is designed to give the user detailed background information so the practice section can have better meaning. This manual is to be used in conjunction with the Simulation Fundamentals Practices manual.

What you've learned

A review of what was covered in the chapter.

CHAPTER 1

Injection Molding Overview

Aim

The aim of this chapter is to review of the injection molding process, flow behavior of thermoplastics in injection molds.

Why do it

Understanding the background of the injection molding process, basic flow behavior of thermoplastics, is critical if the best use of MPI is to be achieved.

Theory and Concepts - Injection Molding Overview

Injection molding

Injection molding is a manufacturing process for making complex shapes from polymeric materials. The process involves heating up the polymer, injecting the molten polymer into a mold, cooling the polymer in the mold, and ejecting the part. There are hundreds of variables involved with the injection molding process. These variables can be categorized into two main groups: machine variables and plastic variables. A machine variable is a variable that can be set or adjusted on the machine. A plastic variable is a variable that affects the plastic directly. For example, barrel temperature is a machine variable, but the actual measured “melt” temperature is a plastic variable. Moldflow Plastics Insight works primarily with plastic variables. Moldflow Plastics Insight concentrates on the behavior of the polymer while it is being injected, cooling, and finally what the shape, or warpage of the part will be when it is ejected from the mold.

Injection molding machine

The injection molding machine has many components. The major ones are listed below:

Hopper

The plastic is put into the machine here in pellet form.

Barrel, Screw

The barrel and screw take the material from the hopper and melt it. This is done primarily by shear heating (screw rotating) but also by conducting heat from the barrel’s heaters. The temperature of the melted plastic is the plastic variable called “melt temperature”. The best method to measure the melt temperature is to purge material from the barrel into a pile of plastic and place a hand held pyrometer probe in the purge pile and wait for the temperature to stabilize.

Hydraulic unit

The hydraulic unit provides the pressure that injects the plastic into the mold. The pressure applied by the oil or hydraulic cylinder is a machine variable. The pressure it produces on the plastic is the plastic variable, “plastic pressure”. The ratio between hydraulic and plastic pressure is called the intensification ratio and is normally about 10:1, but it can be higher.

Clamp unit

The clamp unit holds the injection mold or tool closed while the part is being molded, and opens the mold to eject the part.

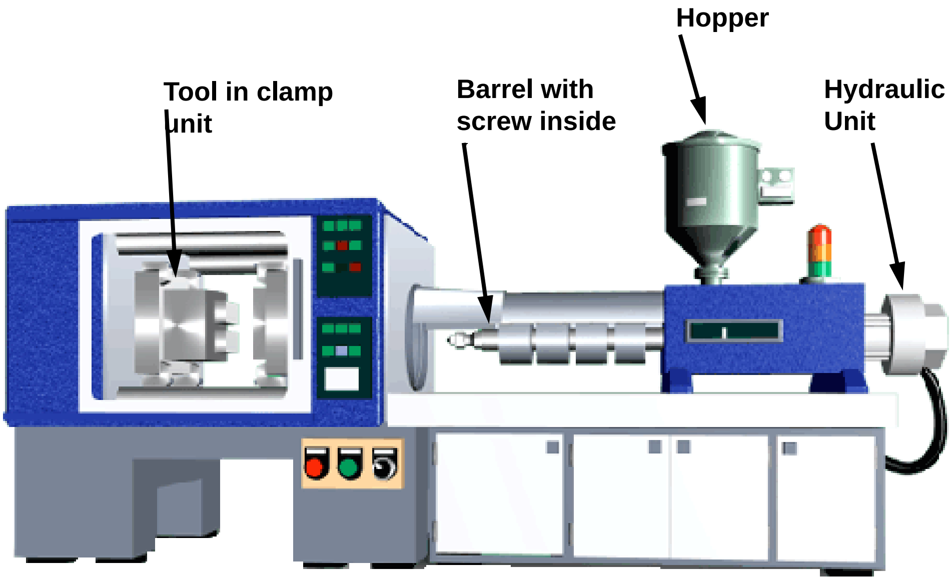


Figure 1: Injection molding machine

Injection molding process

The injection molding cycle has 4 major components, as indicated below:

Fill time

The mold is closed at the start of the fill time. The screw moves forward forcing molten plastic into the mold, which is normally very fast. This part of the process is controlled by velocity. As the plastic hits the cold mold wall it sticks to the wall and freezes. The plastic flow channel is located between the frozen layers of polymer. The rate of injection has a large influence on the thickness of the frozen layer. How the part is filled is the biggest single influence on the quality of the part, in most cases. Filling is normally the smallest portion of the molding cycle.

Hold or Pack time

Once the cavity is filled, the injection molding machine continues to apply pressure to the plastic to pack more material into the mold. This is to compensate for shrinkage as the plastic continues to cool. This part of the process is controlled by pressure. The switch-over from velocity control to pressure control occurs just before the part volume has been filled

Cooling time

There is no longer any pressure applied to the plastic. The part continues to cool and freeze until it is cool enough to eject and hold its shape. While the part is cooling, the screw rotates molten plastic for the next cycle

Mold open time

The mold is opened, the part is ejected, and the mold is closed in preparation for the next shot.

Injection molding cycle

In a typical injection molding cycle, the cooling time is the largest portion of the cycle, followed by hold time, mold open time, and finally fill time, as shown in Figure 2. The cooling time can be over 50% of the cycle time, and the fill time can be around 3% to 5% of the cycle. The relationship between the variables is dependent on the part, material, and machine.

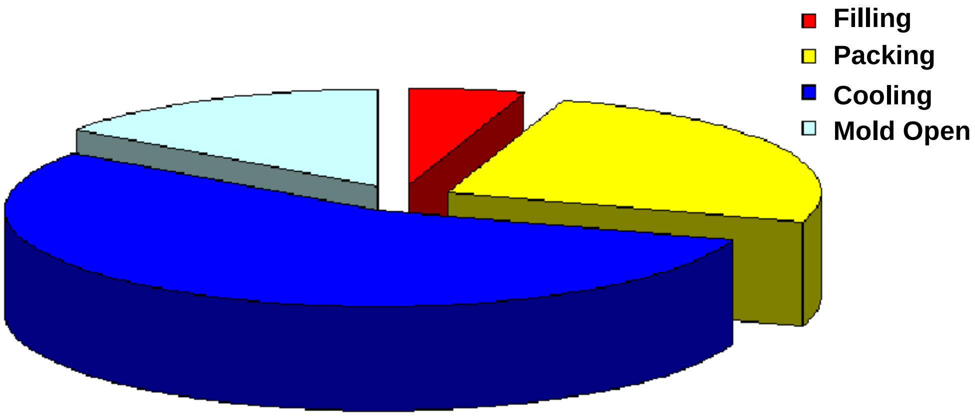


Figure 2: Typical injection molding cycle

Injection mold

The injection mold has two major components: the fixed half and moving half, although other synonyms are used, see Figure 3. For example, the fixed half can also be called the:

- A half.
- Stationary half.
- Cavity.
- Cover.

The moving half can also be called the:

- B-half.
- Moving half.
- Core.

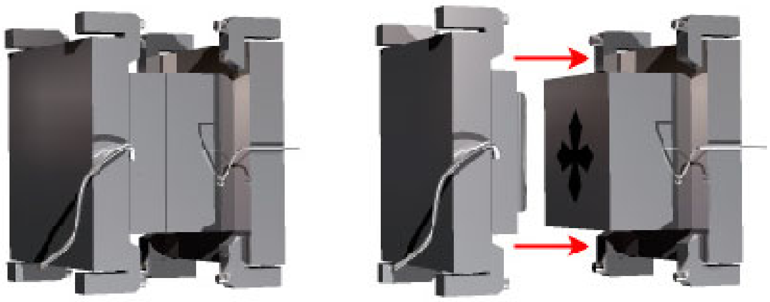


Figure 3: Injection mold

Injection Pressure

Pressure is the driving force that overcomes the resistance of polymer melt, pushing the polymer to fill and pack the mold cavity. If you place a number of pressure sensors along the flow path of the polymer melt, the pressure distribution in the polymer melt can be obtained, as schematically illustrated in Figure 4 below.

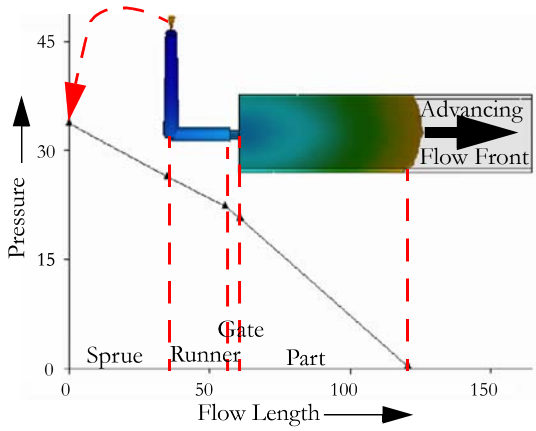


Figure 4: Pressure decreases along the delivery system and the cavity.

Pressure drives the flow front

The polymer flow front travels from areas of high pressure to areas of low pressure, analogous to water flowing from higher elevations to lower elevations. During the injection stage, high pressure builds up at the injection nozzle to overcome the flow resistance of the polymer melt. The pressure decreases along the flow length towards the polymer flow front, where the pressure reaches the atmospheric pressure if the cavity is vented. Broadly speaking, the pressure drop increases with the flow resistance of the melt, which in turn, is a function of the geometry and melt viscosity. The polymer's viscosity is often defined with a melt flow index. However, it is not a good measure of the material's behavior during the filling phase. As the flow length increases, the polymer entrance pressure increases to maintain a desirable injection flow rate.

Factors that influence injection pressure requirements

The following diagrams illustrate the design and processing factors that influence injection pressure.

Table 1: Factors influencing injection pressure

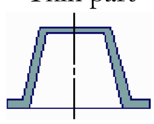
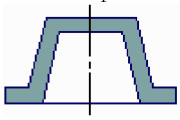
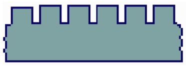
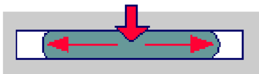
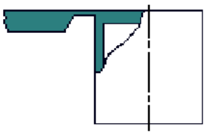
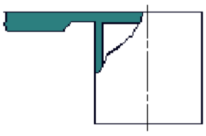
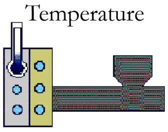
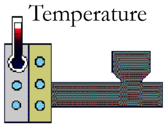
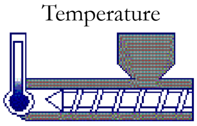
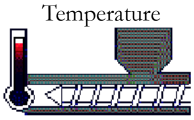
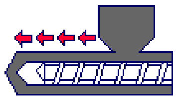
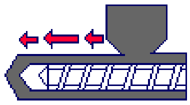
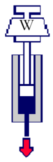
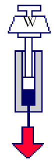
Factor	Variable	Higher injection pressure required	Lower injection pressure required
Part Design	Part thickness	Thin part 	Thick part 
	Part surface area	More Wall Cooling and Drag Force 	Less Wall Cooling and Drag Force 
	Flow Length	Long flow length 	Short flow length 
Feed System Design	Gate Size	Restrictive Gate 	Generous Gate 
	Runner Diameter	Runner Diameter Too Small or Large 	Runner Diameter Optimized 
Processing Conditions	Mold Temperature	Colder Coolant Temperature 	Hotter Coolant Temperature 
	Melt Temperature	Colder Melt Temperature 	Hotter Melt Temperature 

Table 1: Factors influencing injection pressure

Factor	Variable	Higher injection pressure required	Lower injection pressure required
	Ram speed (Injection time)	Improper Ram Speed 	Optimized Ram Speed 
Material Selection	Melt flow index	Low Index Material 	High Index Material 

Flow behavior

Phases of molding

The phases of molding look at the injection molding cycle from a plastic molecule point of view, as indicated below:

Filling phase

As plastic enters the mold, it fills the mold with a controlled velocity. This phase constitutes most of the ram's distance moved during the cycle, as shown in Figure 5. The filling phase ends when the volume of the cavity is just filled. While filling, the material enters the cavity, sticks to the mold wall and freezes. The polymer then fills the part in a fountain flow fashion, at which point a significant amount of shear heat can develop.

Pressurization phase

Once the cavity is filled, the machine ram continues to move forward under pressure control. The ram may continue to move forward for some time based on the compressibility of the plastic. The pressurization phase ends when the maximum pressure occurs, which is at the end of fill. The flow of the polymer is very similar to the filling phase except in regions where the frozen layer is growing rapidly and the flow rate is significantly lower.

Compensation phase

The machine ram, which is controlled by pressure, will continue to move forward. Extra polymer is forced into the mold to compensate for the large volume change that occurs between the molten state of the polymer at the melt temperature, and its solid state at room temperature. Flow in the compensation phase can be very unstable due to temperature instability. This can lead to highly oriented areas in the part, possibly causing warpage.

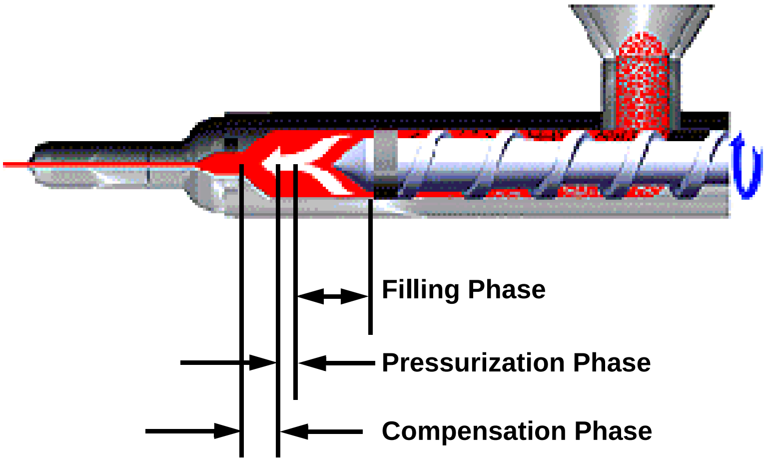


Figure 5: Phases of Injection molding

Fountain flow

Fountain flow describes how the plastic fills the mold. The plastic in the center of the cross section is moving at the highest velocity. When a molecule in the center of the cross section gets to the flow front, it follows the flow front to the mold wall, sticks to it, cools, and forms a frozen layer, see Figure 6. The first molecules injected into the part form the outermost layer of the part, while the later molecules form the center or core of the part.

MPI assumes that the entire filling process is by fountain flow. There may be a short period of time, near poorly designed gates, where fountain flow does not occur. This cannot be analyzed with midplane, Fusion or 3D analysis methods.

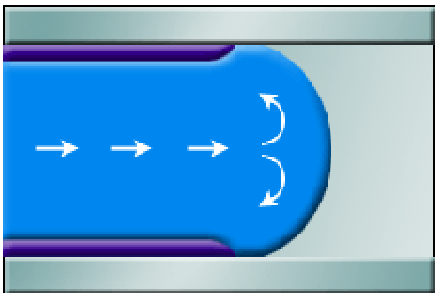


Figure 6: Fountain flow

Cross-sectional flow & molecular orientation

During filling, there is a significant variation in molecular orientation, shear stress, and shear rate distributions through the cross-section of the part. Shear rate is defined as how fast one molecule is sliding past another, or the difference in velocity over distance, measured in units of 1/sec. and called reciprocal seconds. Shear stress is force over an area with units of MPa or psi. While filling the part in a fountain flow fashion, the molecules in the center of the cross section are moving at a high, but relatively uniform, velocity, as shown in Figure 7. Therefore, the shear rate is low and there is very little shearing tensile force on the molecules. Near the frozen layer, the velocity is low, and there is a significant velocity gradient or high shear rate. This creates high shear stress on the molecules, which will stretch or align the molecules in the direction of flow. The curve shown in Figure 8 represents the shear rate in the cross-section. The shear rate is 0 1/sec. in the frozen layer, has a maximum value just inside the layer, and 0 1/sec. in the center of the cross section again.

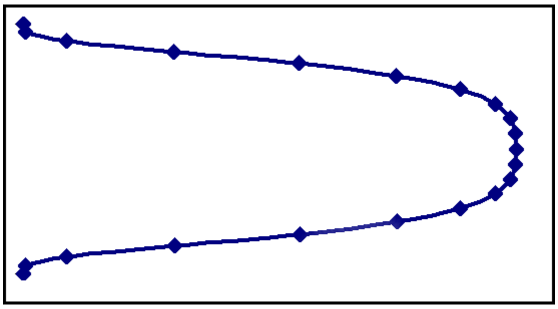


Figure 7: Cross-sectional velocity

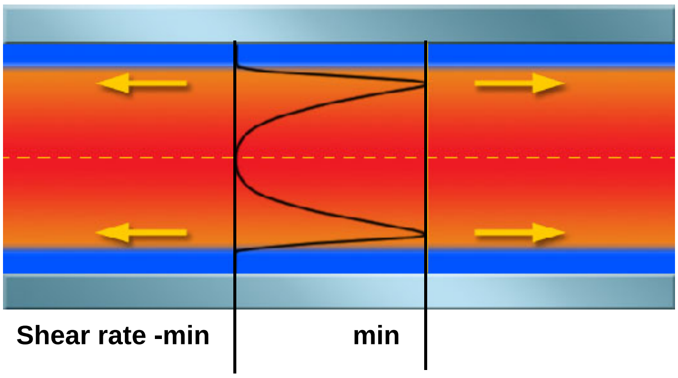


Figure 8: Shear rate through the cross-section

Cross-sectional heat transfer

Due to the high shear rate that occurs just inside the frozen layer, there is a significant amount of heat buildup within the cross section. The center of the flow channel brings fresh hot material from the barrel of the machine. The mold temperature is relatively cold compared to the plastic. When the plastic flows into the mold and hits the mold wall, it sticks, cools, and freezes (see Figure 9). The thickness of the frozen layer stabilizes quickly as long as the flow rate is constant. During filling, most of the heat transfer is from the high shear rate zone just inside the frozen layer to the mold. While the part is filling, the amount of heat extracted into the mold should ideally be equal to the heat generated by shear. Under these conditions, the bulk of the cross section is at a very uniform temperature, which helps promote packing. Temperature uniformity during filling is therefore a primary design guideline when using MPI.

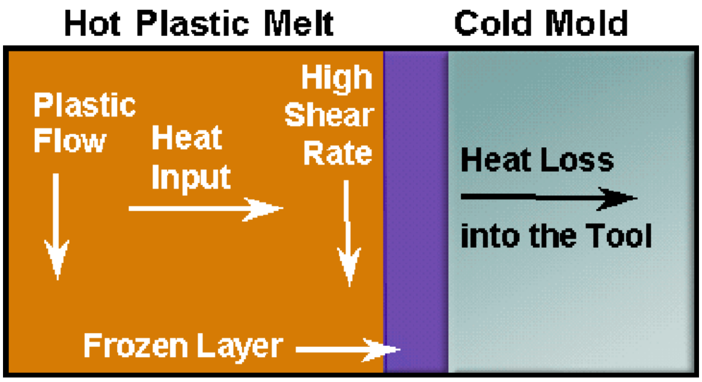


Figure 9: Heat transfer in the cross-section

Injection rate vs. frozen layer thickness

The injection rate has a significant effect on the thickness of the frozen layer during filling. While the part is filling, the amount of shear heat generated is dependent on the fill rate. The higher the fill rate or the shorter the fill time, the more shear heat will be generated and the thinner the frozen layer will be. The slower the injection rate, the thicker the frozen layer. As the frozen layer gets thicker, the actual flow channel for the plastic gets smaller and therefore a higher fill pressure is required for a given flow rate. The opposite is true when the frozen layer gets thinner, as represented in Figure 10.

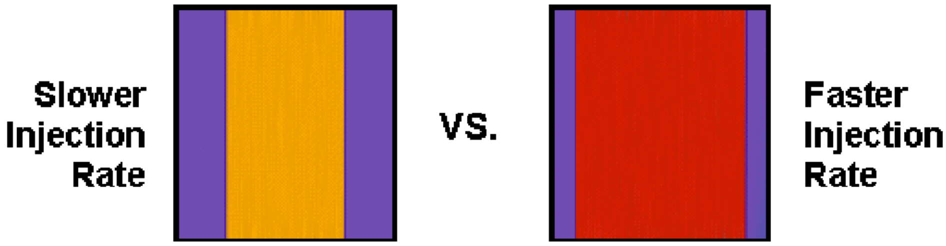


Figure 10: Effect of injection rate on frozen layer

The graph in Figure 11 below represents fill times from 0.2 seconds to 20.0 seconds. The part has a flow length of about 900 mm with a nominal wall thickness of 3.0 mm. The material is ABS, and the X-axis is the fill time required to fill the part. At very fast fill times, the pressure to fill is dominated by shear, or the rate of injection. As the fill time gets longer, the pressure goes down. At a fill time of about 5 seconds, the pressure starts to rise again because the plastic is loosing temperature and the viscosity of the material is getting higher. Heat transfer rather than flow dominates the pressure after about 5 seconds. All thermoplastics and all parts exhibit this “U” shape behavior. The shape of the “U” may change, and the time scale will change, but the pressure vs. time curve will always exhibit this characteristic “U” shape. The Upper limit is equal to the melt temperature entering the part, and the lower limit has a 20ºC temperature drop in the part.

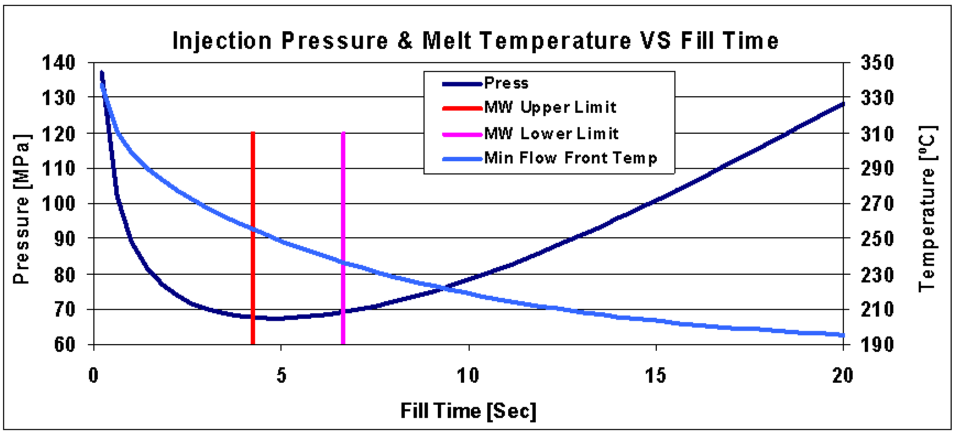


Figure 11: Injection pressure & melt temperature vs. fill time

Pressure-Volume-Temperature (PVT)

PVT data is used to model the compressibility of a material. This information is vital for packing and also important for the filling of the part(s). The graph below in Figure 12 shows PvT graphs for both amorphous and semi-crystalline materials. For both amorphous and crystalline materials, the basic trend is the same: as the material cools down and as the pressure applied gets higher, the specific volume gets lower and the density gets higher. Even during the filling phase, the melt is compressed while the plastic is flowing. The difference between the curves is the knee in the curve of the Semi-Crystalline material. When the crystalline structure forms, there is a rapid reduction of specific volume (increase in density).

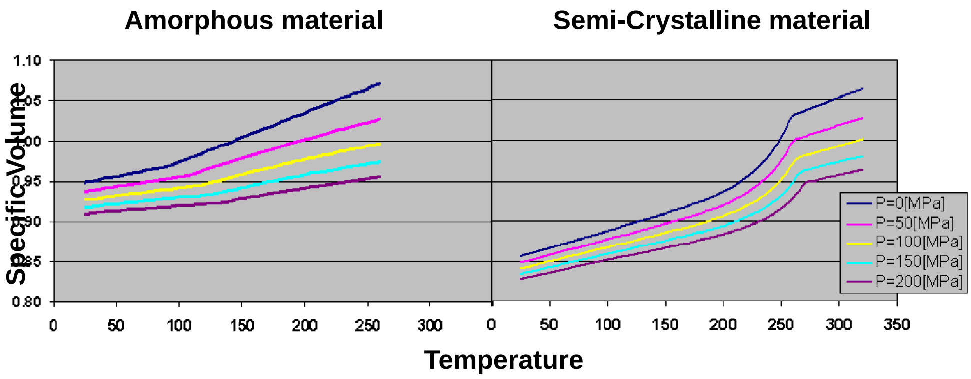


Figure 12: Pressure-volume-Temperature (PvT) data for an amorphous and semi-crystalline material

Shrinkage

The shrinkage behavior of thermoplastic materials is quite complex. Generally, there is more shrinkage in the direction of flow than across the direction of flow for unfilled materials. When glass fiber fillers are added, the trend is reversed and there is more shrinkage across the flow direction. In Figure 13, both parts were gated with a gate along the entire left edge of the part. They were both made with the same material and used the same processing conditions. The difference is the bottom part used the fiber flow analysis to take into account the orientation of the glass fibers. Without considering the glass fibers, there is more shrinkage parallel to flow. When considering the effect of the glass fibers, there is much less shrinkage. The shrinkage perpendicular to flow is less than the shrinkage parallel to flow. The processing conditions used have a significant effect, as does the type of polymer. The warpage analysis uses data from the filling and packing analysis to determine the exact amount of shrinkage.

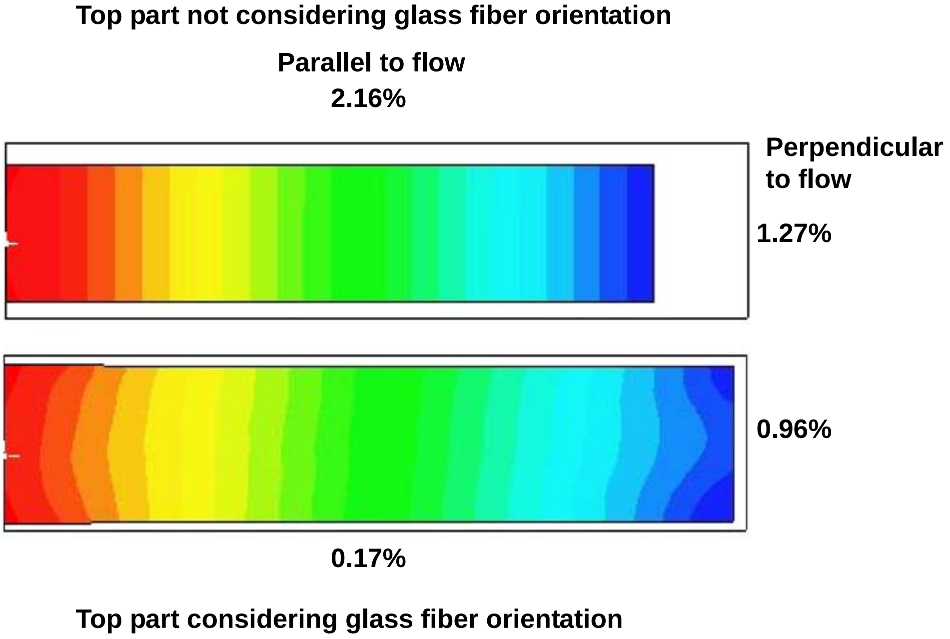


Figure 13: Shrinkage of a plaque considering fiber orientation and not considering orientation

What You've Learned

The injection molding machine has the 4 major components including:

- Hopper.
- Barrel, Screw.
- Hydraulic unit.
- Clamp unit.

The injection molding process is comprised of 4 main components making up the cycle time including:

- Fill time.
- Hold or Pack time.
- Cooling time.
- Mold open time.

The mold cooling time is normally the largest part of the cycle time, and fill time normally has the most influence on the part quality.

The injection mold has two major components: the fixed half and moving half.

The injection molding machine with it's hydraulic unit produces pressure the is used during the filling and packing times. Pressure drives the flow front advancement. The following parameters influence pressure requirements:

- Part thickness.
- Part surface area.
- Flow Length.
- Gate Size.
- Runner Diameter.
- Mold Temperature.
- Melt Temperature.
- Ram speed (Injection time).
- Melt flow index.

The pressure created by the molding machine impacts the polymer. Control of the polymer is done indirectly by the machine settings. What happens to the polymer in the mold is divided into phases of molding which include:

- **Filling phase** - Velocity controlled filling of the entire cavity volume.
- **Pressurization phase** - Building up of pressure in the mold to a maximum value.
- **Compensation phase** - Continuing to apply pressure to compensate for thermal shrinkage.

Polymers being molded have the following behaviors:

- **Fountain flow** - The first material in forms the skin of the part and the last material in forms the core.

- **Molecular orientation** - Due to fountain flow, the highest degree of molecular orientation and shear rate is just inside the frozen layer and is very low in the center of the cross section.
- **Heat transfer** - The high shear rate inside the frozen layer develops shear heat. Most of this heat is transferred to the mold wall during filling. There should be a balance between shear heat developed and the heat transfer to the mold.
- **Injection rate vs. frozen layer thickness** - The faster the injection rate, the more shear heat will occur, and the thinner the frozen layer will be.
- **Pressure-Volume-Temperature (PvT)** - The PvT of the polymer determines how compressible the material is. This has a significant influence on the packing of the part.
- **Shrinkage** - Unfilled polymers normally shrink more parallel to the flow direction. Fiber filled materials tend to shrink less than unfilled materials, but shrink more perpendicular to flow.

CHAPTER 2

Finite Element Overview

Aim

The aim of this chapter is to review the finite elements used by MPI and how they are combined into different meshes. Also to review the assumptions used by the flow solvers for the different element types.

Why do it

The 3 mesh types used by MPI all have their advantages, disadvantages and assumptions. When there is a choice in the mesh type to be used, it is critical the choice is made based considering these advantages, disadvantages and assumptions.

Overview

There are three types of finite elements used by MPI including:

- 2-noded beam.
- 3-noded triangle.
- 4-noded tetrahedral.

These 3 element types are used in different combinations to create different "meshes" used by MPI, including:

- Midplane.
- Fusion.
- 3D.

The elements and meshes have different flow solver assumptions that will in part determine what mesh is best to use. These assumptions will be reviewed.

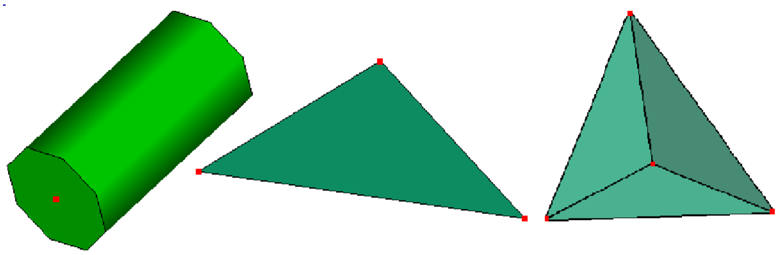
Theory and Concepts - Finite Element Overivew

Finite elements used in Moldflow

In order to run a Moldflow analysis, the part must have an appropriate finite element mesh created. Often, the finite element mesh is referred to as a **mesh**. Elements divide the geometry of the part (domain), or other tool component into a number of small domains. These small domains or elements are defined by nodes, (coordinates in space) and are used for the calculations inside Moldflow. There are three main categories of elements:

- Beam.
  - 2-noded element used to describe the feed system, cooling channels etc.
- Triangle.
  - 3-noded element used to describe the part, mold inserts etc.
- Tetrahedral.
  - 4-noded element used to describe the parts, cores, feed systems etc.

An example of these 3 element types are shown in Figure 14.



Beam Element **Triangular Element** **Tetrahedral Element**

Figure 14: Element Types

Mesh types used by Moldflow

Moldflow uses three "mesh types" for analysis. The mesh types use a combination of the element types described above. The mesh types are:

Midplane

- The mesh is defined on the "midplane" or center line of the plastic cross section as shown in Figure 15 A.
- Triangular elements are primarily used to define the part.
- Beam elements can be used to define the feed system, cooling channels etc.

Fusion

Triangular elements are defined on the surface of the plastic as shown in Figure 15 B.

- Analysis method called Dual Domain™.
- Beam elements can be used to define the feed system, cooling channels etc.

3D

- Tetrahedral elements are used to represent the part. Several rows of elements are used to define the cross section as shown in Figure 15 C.
- Beam elements or tetrahedral elements can be used to represent the feed system.

Care should be used when using the term "mesh". Depending on the context, it could be referring to a collection of a certain type of finite element, a "triangular mesh" or it could mean a type of analysis, "A midplane mesh was used".

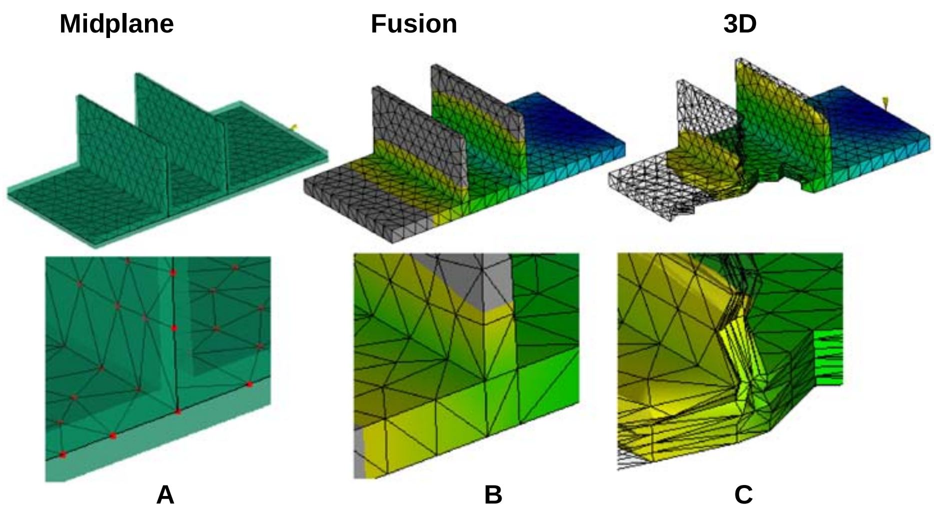


Figure 15: Mesh types

Solver assumptions

Midplane and Fusion

The same flow solver is used for midplane and Fusion mesh types. Every type of solver has certain assumptions. For midplane and Fusion the solvers are based on the generalized Hele-Shaw flow model. This model has the assumptions listed below:

- Laminar flow of a generalized Newtonian fluid.
- Inertia and gravity effects can be ignored.
- In plane heat conduction is negligible compared to conduction in the thickness direction.
- Thermal convection in the thickness direction is neglected.

- Heat loss from edges can be ignored for the triangular element type.

Element specific assumptions

Beams

Sometimes referred as a **1D element,** it has an assigned cross sectional size and shape. Beams are axisymmetric circular-tube flow of a generalized Newtonian fluid. A non-circular shape typically is represented by an equivalent circular tube with the same hydraulic diameter but with the volumetric flow rate scaled down in order to give the same average velocity as the original shape. Juncture losses due to abrupt contractions in flow path are incorporated through an empirical model derived based on Bagley corrections in viscosity characterization. Beam elements can not account for shear-induced imbalances as sometimes seen in feed systems.

Triangles in midplane meshes

A triangular element used in a midplane mesh is often referred to a 2.5D element or shell element. This mesh simulates a 3D part with a 2D plane surface at the center of the thickness. A thickness property is assigned to this plane hence the terminology 2.5 D. Because of the assumptions listed above, the cross section that can be modeled with this element type is limited. As a minimum, the width to thickness ratio of any local area should be at least 4:1, otherwise significant errors may be introduced. At a 4:1 width to thickness ratio, 20% of the perimeter is in the thickness direction and is not accounted for in the heat transfer equations. The greater the violation of this rule; the greater the amount of possible error. This is a particular problem for square shaped geometry such as connecting ribs, housing vents or grills.

Triangles in Fusion meshes

Fusion meshes sometimes called a modified 2.5D mesh, simulates a 3D part with a boundary or skin mesh on the outside surfaces of the part. The main difference between midplane and Fusion meshes is how the thickness is determined. In Fusion, elements across the thickness are aligned and matched. The distance between the mesh on the opposite side of the wall defines the part thickness. The mesh density is an important factor in determining the accuracy of the thickness representation, in particular on tapered features such as ribs. The percentage of the elements in the Fusion mesh that are matched is a key factor in determining the quality of the mesh. It should be at least 85%.

3D meshes

A 3D mesh makes fewer assumptions than Midplane and Fusion including:

- Uses full 3D Navier-Stokes.
- Solves for pressure, temperature and the three directional velocity components at each node.
- Considers heat conduction in all directions.
- Provides options to use inertia and/or gravity effects.

3D meshes create a true 3D representation of the part. A 3D mesh works well with “thick and chunky” parts that violate the thickness rules stated previously such as electrical connectors and thick structural components.

What You've Learned

The elements used in MPI are:

- 2-noded beams used to describe the feed system, cooling channels etc.
- 3-noded triangular elements used to describe the part, mold inserts etc.
- 4-noded tetrahedral elements used to describe the parts, cores, feed systems etc.

The meshes used in MPI include:

- Midplane, using triangular elements for the part and beam elements for the feed system.
- Fusion, using triangular elements for the part and beam elements for the feed system.
- 3D, using tetrahedral elements for the part and feed system, and optionally beams for the feed system.

Midplane and Fusion use the Hele-Shaw model. Most of the assumptions related to the Hele-Shaw model relate to limiting the heat transfer. As a result, the thickness to width ratio of the flow path should be at least 4:1.

3D uses a Navier-Stokes model which considers heat transfer in all directions, and can account for inertia and gravity which midplane and Fusion cannot.

CHAPTER 3

Moldflow Design Principles

Aim

The aim of this chapter is to review the Moldflow Design Principles to be used with MPI.

Why do it

MPI was designed in part to analyze the molding issues addressed in the Moldflow Design Principles. Taking into account Moldflow Design Principles will reduce problems will part and mold design and will make the part easier to mold.

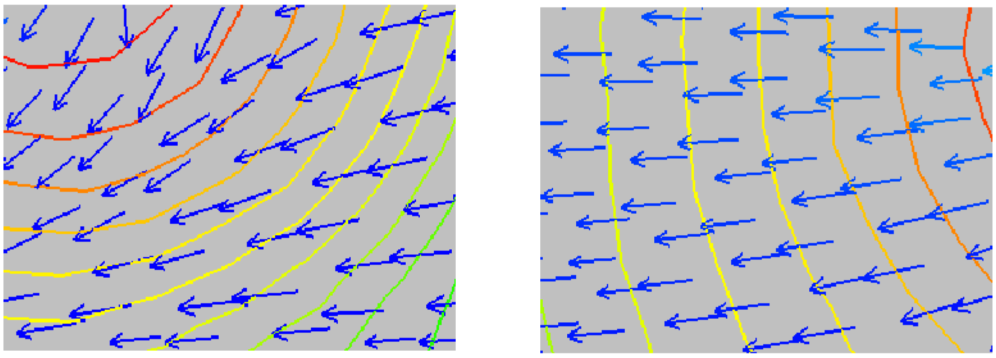
Theory and Concepts - Moldflow Design Principles

The Moldflow Design Principles are a set of rules that influence the design of a part and tools to optimize the filling of a part. When these principles are followed, higher quality parts and faster cycle times are the result. Not following the principles leads to problematic designs. The design principles include:

- Unidirectional and controlled flow pattern.
- Flow balancing.
- Constant pressure gradient.
- Maximum shear stress.
- Uniform cooling.
- Positioning weld and meld lines.
- Avoid hesitation effects.
- Avoid underflow.
- Balancing with flow leaders and flow deflectors.
- Controlled frictional heat.
- Thermal shutoff for runners.
- Acceptable runner/cavity ratio.

Unidirectional and controlled flow pattern

To produce unidirectional orientation, the filling pattern in the part should be unidirectional. This means the flow direction should be straight and not reverse on itself during filling. In the examples below in Figure 16, the part is rectangular. In the example on the left, the part is gated near the center, and the flow changes direction during filling. It does not have unidirectional orientation. In the example on the right, the part is gated on the right edge and so the flow front lines and flow direction arrows are perpendicular, producing unidirectional orientation.



Center Gated
Flow not Unidirectional

End Gated
Flow Unidirectional

Figure 16: Flow pattern changes with gate location

Flow balancing

All the flow paths within a mold should fill at the same time and with equal pressure. For multi-cavity molds, this means each cavity should fill at the same time. Within parts, the same holds true; the extremities of the part should fill at the same time also.

There are two types of balanced runner systems: naturally balanced, sometimes called geometrically balanced runners, and artificially balanced runners. In a naturally balanced runner system, the flow length from the sprue to each of the parts is the same for all cavities, as shown in Figure 17. Generally this type of runner system has a larger processing window than artificially balanced runners.

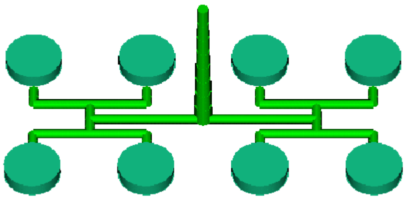


Figure 17: Naturally balanced runner system

The artificially balanced runner system achieves its balance by changing the size of the runners. This can be a very useful technique for balancing runners, as there is generally less runner volume required than a naturally balanced runner. However, due to the changes in the runner diameter, the processing window is generally smaller than a naturally balanced runner. Injection time is generally the main limiting factor. As a result, in situations that have high runner pressure drops plus low part pressure drops, tight tolerances, thin sections in the part, and potential sink mark issues, artificially balanced runners may have a very small molding window or may not be practical. The greater the runner length ratio between the longest and shortest path the greater the potential problem. Figure 18 shows an example of an artificially balanced runner.

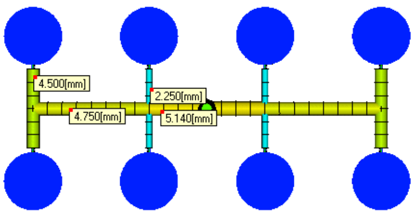


Figure 18: Artificially balanced runners

Constant pressure gradient

The pressure gradient while the part is filling should be uniform through the part. Figure 19 shows a part that does not have a constant pressure gradient during filling. The XY graph is the pressure at the injection location. Just at the beginning of fill, there is a spike in pressure. However, the big problem is at the end of fill. The part is mostly filling by radial flow. As the flow front meets the center of the sidewalls the flow front starts contracting. This corresponds to a slight increase in the pressure gradient. The big spike occurs when the three corners fill and the remaining upper right corner is the only area that remains unfilled. All the material exiting the gate enters the upper right corner causing the pressure spike. The volumetric flow rate entering the part is constant. The pressure gradient is an indication of a balance problem, or it suggests an injection velocity profile should be used.

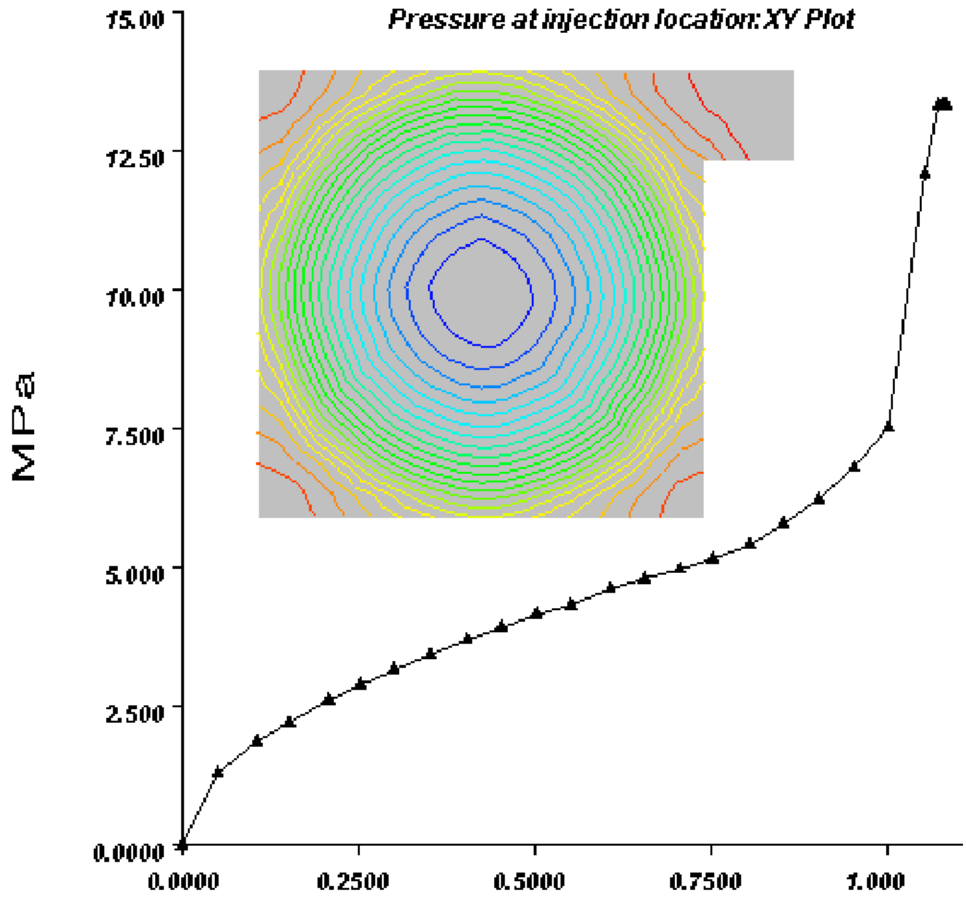


Figure 19: Pressure Gradient

Maximum shear stress

The maximum shear stress in the part should be below the material limit specified in the material database. The shear stress limit is approximately 1% of the tensile strength of the material and is also application-specific. For parts used in harsh environments such as elevated temperatures, under a high load during use, or exposed to chemical attack, the limit specified in the database may be too high. Alternatively, if the part is not used in a harsh environment the limit is conservative (low), and the stress can be significantly exceeded without any problems. However, when the shear stress does get above the limit, it should be kept as low as possible. Figure 20 shows the maximum shear stress in the part scaled from the material limit to the maximum shear stress value calculated in the analysis. Areas that are colored in the plot are therefore above the limit. In this case, the maximum shear stress is 0.45 MPa which is not too high. Most of the time, parts will have areas of high shear stress that will be 2 to 5 times the stress limit. In this case, it is only 1.5 times the limit. However, much of the part is slightly about the limit. The maximum shear stress in the cross section is at the frozen/molten layer interface, or "at wall".

Three main factors influence shear stress, including:

- **Wall thickness** - increase the wall thickness to reduce stress.
- **Flow rate** - lower the flow rate (locally or globally) to reduce stress.
- **Melt temperature** - increase the melt temp. to lower the shear stress.

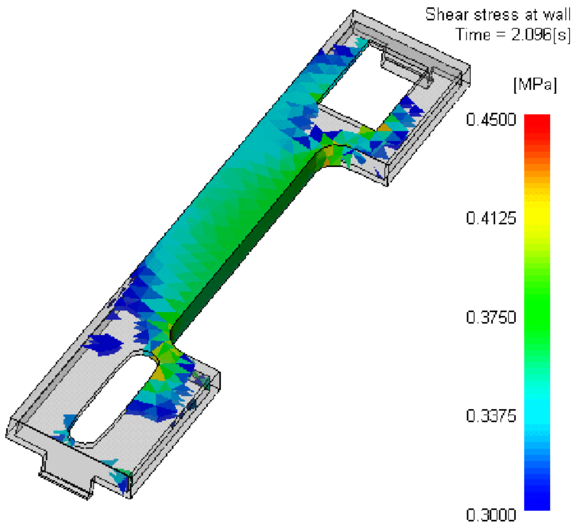


Figure 20: Maximum shear stress

Uniform cooling

When cooling a part, the mold surface temperature should be uniform on both sides of the part. When the temperatures are not uniform, the molecules on the hot side have a longer time to cool so they shrink more. This makes them shorter, so the parts will bow towards the hot side of the part as shown in Figure 21. The left image shows the temperature distribution and the right shows the warpage.

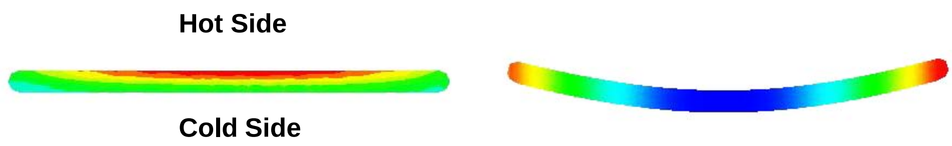


Figure 21: Cooling uniformity

Figure 22 shows a typical "Box" type structure that many injection molded parts have. In the box structure, there is an inside corner (the core) that is normally difficult to cool and where heat tends to concentrate. On the other hand, the cavity side is easy to cool, and there is a larger volume of mold to absorb the heat from the plastic. As a result, the inside of the corner runs hot, allowing more time for the molecules to cool down and shrink therefore collapsing the corner a bit. This will pull the sides of the box towards the core.

Figure 22: Cooling of box structures

Positioning weld and meld lines

A weld line is formed when two flow fronts meet head-on. A meld line is formed when the flow fronts meet while flowing in the same direction. Formation of weld lines and meld lines are shown in Figure 23. Weld lines are generally weaker and more visible than meld lines, but they both should be avoided.

Every time a gate is added to the part, an additional weld or meld line is formed, and so eliminating extra gates is advisable. When the number of weld or meld lines cannot be reduced, they should be placed in the least sensitive or critical areas with regards to their strength and appearance. Depending on the application, a weld or meld line could be a problem in terms of either strength or appearance. The strength of a weld or meld line is generally improved when they are formed at higher temperatures and when the pressures to pack them out is higher.

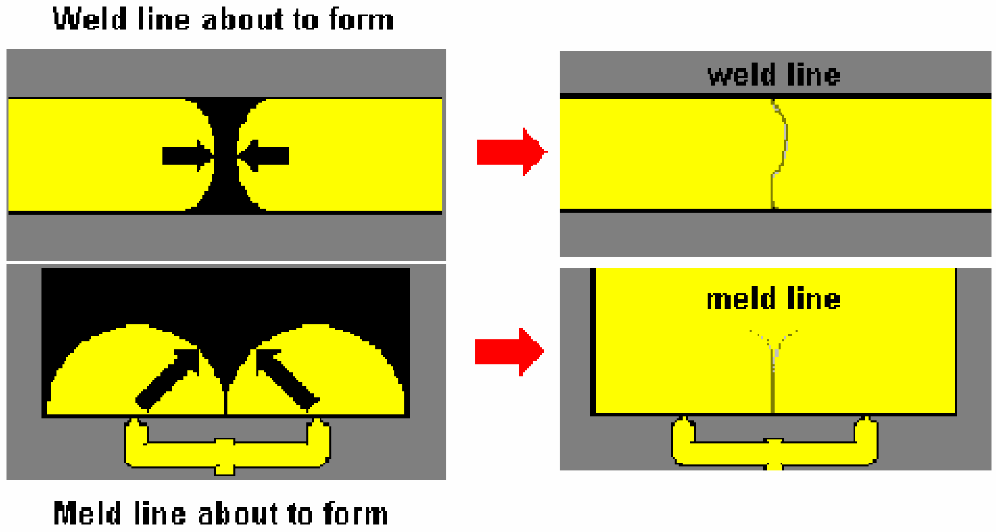


Figure 23: Weld lines and meld lines

Avoid hesitation effects

Hesitation is an unintended slowing down of the flow front. When a flow front slows down too much, it gets too cold and in severe cases can freeze off. This is what has happened in the top example in Figure 24. Hesitation will occur when there is a large variation in wall thickness in the part. In this case, the rib is much thinner than the nominal wall. Having a fast injection time can minimize hesitation. This increases shear heating and provides less time for the material to hesitate. Another way to reduce hesitation is to gate as far as possible from thinner areas, as was done in the bottom example.

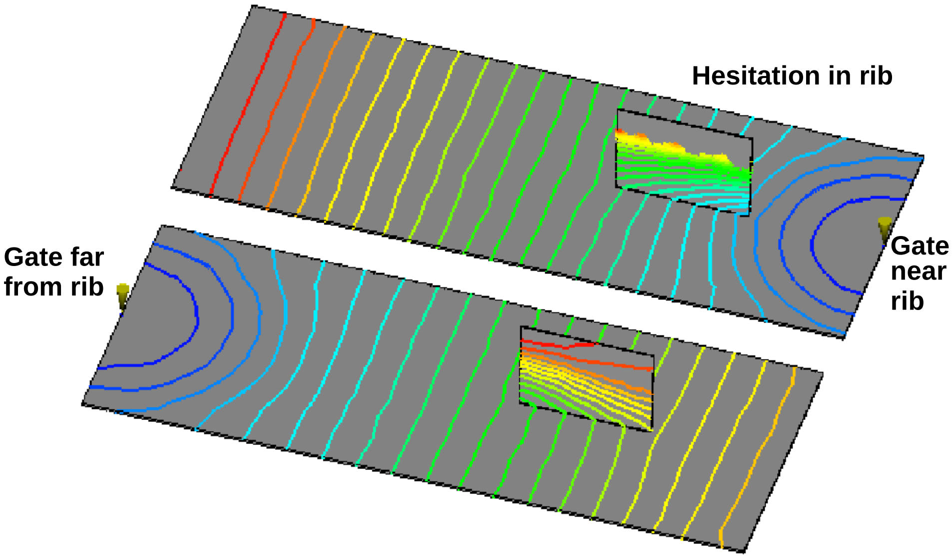


Figure 24: Hesitation caused by gate placement

Avoid underflow

Underflow occurs when a flow front changes direction during filling. In the example in Figure 25, underflow occurs because the flow front is not balanced due to the gate location. The contour lines and the velocity arrows should be perpendicular as they are in the upper left corner of the figure. In the lower right side of the figure they are parallel, indicating a significant shift in the flow direction.

The problem with underflow is its effect on orientation. The initial filling direction for an area on the part is represented by the fill time contours. The flow direction is perpendicular to the contour line. The molecules are initially oriented in the direction of that flow. If, later on during the filling phase, the flow direction changes, the molecules closer to the center of the flow channel are oriented in the new flow direction. Molecules want to generally shrink more in the direction of orientation, and so if there is underflow there is significant internal stress in the location of the underflow.

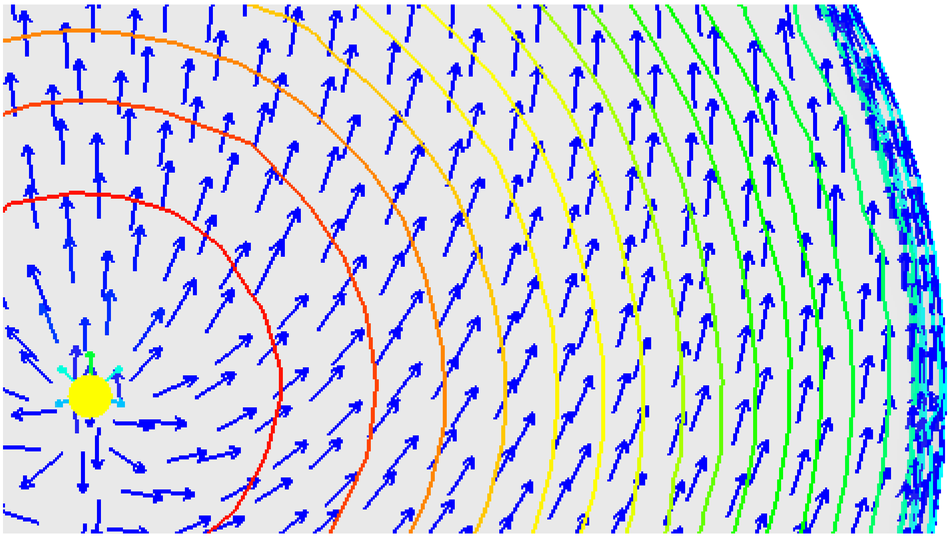


Figure 25: Underflow

Balancing with flow leaders and flow deflectors

Flow leaders are local increases in the nominal wall thickness, whereas flow deflectors are local decreases in thickness.

Many times a part cannot be balanced by gate placement alone. It can be useful to slightly change the wall thickness to enhance or retard the flow in a certain direction. This will allow the filling of the part to be balanced, even though the flow lengths from the gate to the extremities of the part are not equal (see Figure 26). Generally it is better from a material saving point of view to decrease the wall thickness. This may not always be possible, however, due to structural requirements of the part.

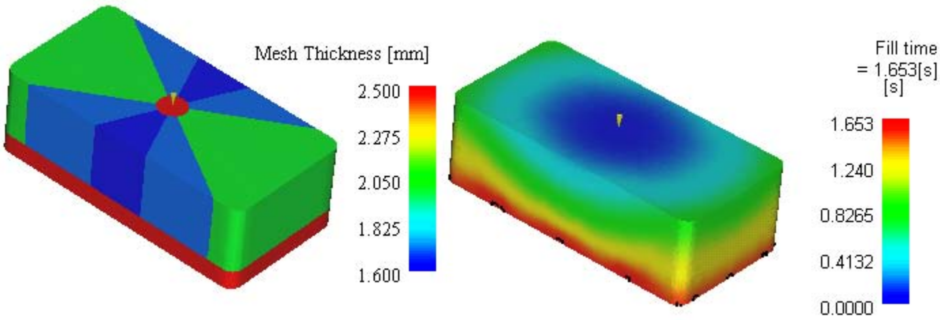


Figure 26: Flow leaders

Controlled frictional heating

Runners should be sized so there is shear heat in the runner. For most materials, 20ºC is good target for the amount of shear heat. Some materials can have a much higher level of shear heat. Materials like PVC should keep shear heat to a minimum.

Shear heat in runners is advantageous for several reasons because it:

- Reduces the pressure to fill the part by increasing the melt temperature entering the part.
- Reduces the shear stress in the part because the melt temperature is higher.
- The barrel melt temperature can be lower resulting in a longer residence time before the material degrades.

The amount of shear heat is controlled by the size of the runners. The smaller the runner, the more shear heat. Following best analysis practices, the melt temperature is optimized by considering only the part initially. This allows you to design the part to be molded at higher temperatures to achieve the benefits mentioned above. When the runners are sized, the melt temperature entering the sprue is lowered so by the time the material enters the part it is back to the optimized temperature.

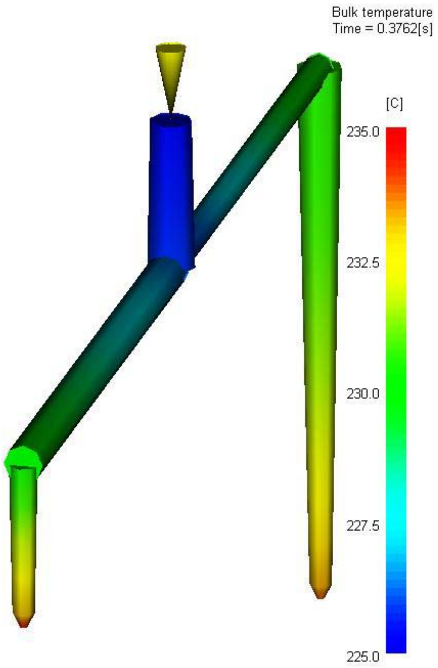


Figure 27: Runner with frictional heat

Thermal shutoff of runners

The runners should be sized so they allow the parts to fill and pack out without controlling the cycle time. In Figure 28, the freeze time of the part is about 3.4 seconds. The runners have freeze times that are at least that of the part. Notice, however, that the cooling time of the sprue is about 10 times that of the part. This would suggest the sprue is too large and should be made smaller if possible. The largest cooling time in a runner should preferably be at most 2 to 3 times that of the part, but this is often difficult to do. In the case of the runner in Figure 28, if the runners were made smaller while maintaining a balanced runner system, the smallest runner, which currently has a cooling time of 4.7 seconds, would quickly become much smaller than the part. As a general rule, if there are no critical dimensions or sink mark quality criteria, the cooling time of the runners can be as low as about 80% of the cooling time of the part. When dimensions are more critical, the cooling time for the runners should be greater than the part.

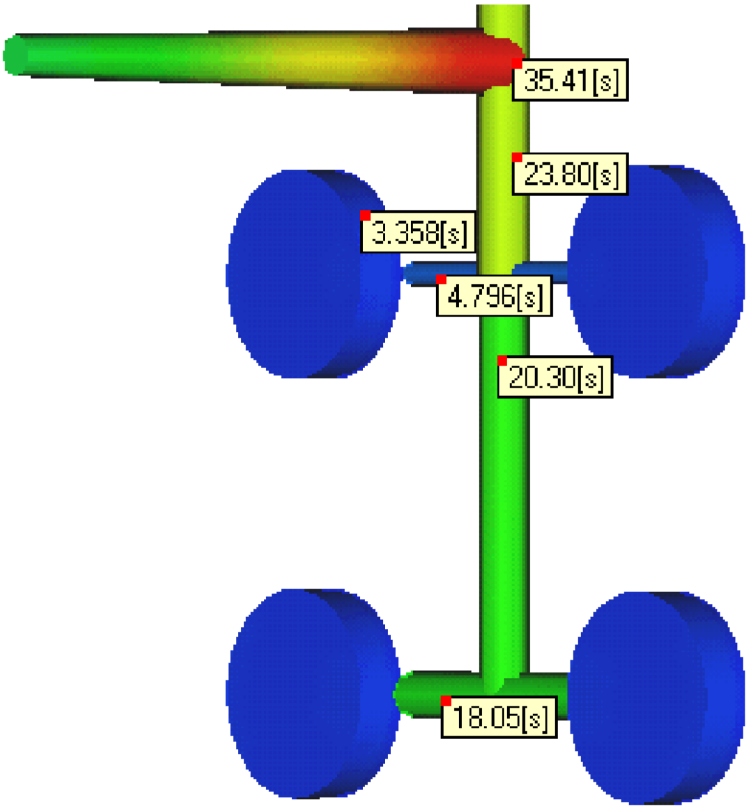


Figure 28: Thermal shutoff

Acceptable runner / cavity ratio

The ratio of the volume of the feed system to the total volume of the cavities should be as low as possible. This is to reduce the material being wasted in the runners and to reduce the amount of regrind. In Figure 29, the runners cannot be made much smaller and still maintain a balanced fill and acceptable packing. In this example, the ratio of runner to cavity volume is 85%, which is very high. Ideally, the volume of the runners should be 20% of the part volume or less.

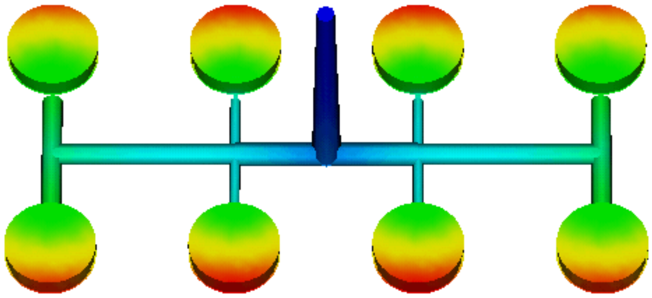


Figure 29: Cavity to runner ratio

What You've Learned

Flow analysis should be used in conjunction with recognized design principles to optimize the part. These design principles include:

- **Unidirectional and controlled flow pattern**. The flow direction should be straight and not reverse on itself during filling.
- **Flow balancing**. All the flow paths within a mold should fill at the same time and with equal pressure.
- **Constant pressure gradient**. The pressure gradient while the part is filling should be uniform through the part.
- **Maximum shear stress**.The maximum shear stress in the part should be below the material limit specified in the material database.
- **Uniform cooling**. When cooling a part, the mold surface temperature should be uniform on both sides of the part.
- **Positioning weld and meld lines**. When the number of weld or meld lines cannot be reduced, they should be placed in the least sensitive or critical areas with regards to their strength and appearance.
- **Avoid hesitation effects**. Hesitation is an unintended slowing down of the flow front and should not occur.
- **Avoid underflow**. Underflow occurs when a flow front changes direction during filling and should not occur.
- **Balancing with flow leaders and flow deflectors**. Flow leaders/deflectors can be useful to slightly change the wall thickness to enhance or retard the flow in a certain direction to balance the part.
- **Controlled frictional heat**. Runners should be sized so there is acceptable amount of shear heat in the runner.
- **Thermal shutoff for runners**. The runners should be sized so they allow the parts to fill and pack out without controlling the cycle time.
- **Acceptable runner/cavity ratio**. The ratio of the volume of the feed system to the total volume of the cavities should be as low as possible.

CHAPTER 4

Introduction to Synergy

Aim

To learn the many features of the MPI/Synergy user interface.

Why do it

The MPI/Synergy graphical user interface provides you with a quick, easy-to-use method of preparing, running and post-processing an analysis of a particular part design. This chapter will introduce you to the various aspects of the user interface and show you how they can be used, as well as getting you up and running with a project and interacting with a model.

Overview

In this exercise, you will review and use the many interface features provided in MPI/Synergy, including:

- Menus
- Panel, Project pane
- Panel, Study tasks
- Panel, Tools
- Layers
- Toolbars
- Context menu
- Display window
- Preferences
- Toolbars
- Working with projects
- Entity selection
- Properties
- Model manipulation
- Wizards

Theory and Concepts - Introduction to Synergy

What is MPI?

Moldflow Plastics Insight® (MPI®) is a product suite designed to simulate the plastic injection molding process and its variants such as gas-assist injection molding, injection-compression, thermosets processing, etc. MPI consists of a single, common user interface called MPI/Synergy and a range of analysis products. Together they provide an insight into the many and varied aspects of plastic injection molding. It is therefore an essential tool for a wide range of users.

What is MPI/Synergy?

MPI/Synergy, also known as just **Synergy**, is the graphical user interface for MPI. It provides a quick, simple method of preparing, running and post-processing an analysis for a model. It also has fast, easy-to-use wizards for creating multiple cavities, runner systems, cooling circuits, mold boundaries and inserts. Included with MPI is a material searching capability for the extensive material database. Material creation tools exist to import, change/modify and create materials to be used for any MPI analysis. To communicate your results with colleagues, MPI has a report generation facility that creates reports. You can customize the reports to contain any of the results derived from an analysis. The reports can contain images of the part(s) analyzed, including any of the animated results. One report can contain results from any number of analyses.

MPI/Synergy is a fully integrated solution for all MPI analysis modules including;

Table 2: MPI modules and supported mesh types

Module	Midplane	Fusion	3D
MPI/Flow	X	X	X
MPI/Cool	X	X	X
MPI/Warp	X	X	X
MPI/Fiber	X	X	X
MPI/Stress	X		
MPI/Gas	X		X
MPI/Optim	X	X	
MPI/Co-Injection	X		
MPI/Injection Compression	X		
MPI/Reactive Molding	X	X	X
MPI/Microchip Encapsulation	X		X
MPI/Underfill Encapsulation	X		X
MPI/Mucell^{tm}	X	X	

Starting MPI/Synergy

MPI/Synergy is started on a PC two ways. During installation, there is a shortcut created that looks like  with the description **Plastics Insight** with a version number. Synergy can also be started by the start menu. The path will be something like, **Start ➨ All programs ➨ Moldflow Plastics Insight ➨ Plastics Insight**.

The MPI/Synergy graphical user interface

MPI/Synergy, shown in Figure 30 is the environment used to prepare, run and post-process an analysis for all MPI analysis modules. Within Synergy, you can set up a sequence analysis (such as cool-flow-warp), run the analysis, view the results and prepare a report.

There are six main sections of MPI/Synergy:

- Menus.
  - ➤ Provides quick access to the functionality of MPI.
- Panel.
  - ➤ Area on the side of the screen that contains panels used for project, study, and layer management plus tools used for geometry creation, mesh diagnostics and mesh cleanup.
- Toolbars.
  - ➤ Series of icons that allow for quick access to most commands in Synergy.
- Context menu.
  - ➤ Menu accessed with a right click that is context sensitive. Different options appear depending on where the cursor is when the context menu is activated.
- Display.
  - ➤ This is the area of Synergy where documents (studies) are opened.
- Wizards.
  - ➤ A wizard is a tool that helps you preform a specific multi step task

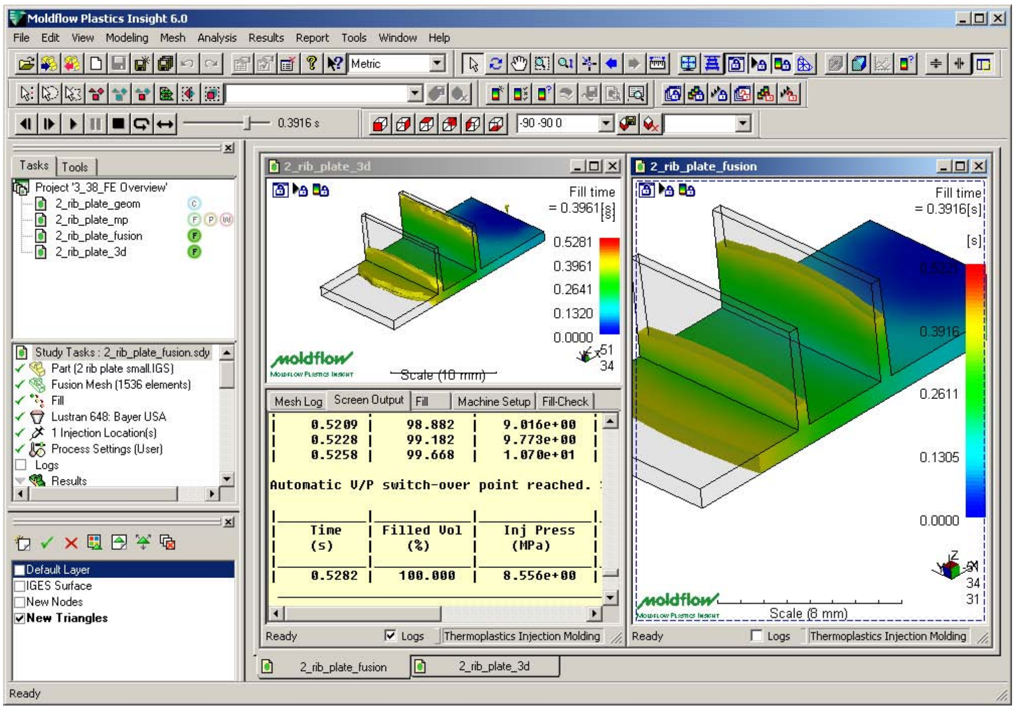


Figure 30: MPI/Synergy window

Menus

The menus at the top of the MPI/Synergy window contain the commands available in MPI. The commands available in the menus depend on the stage of the simulation process you are currently in. Table 3 below describes the function of each menu.

Table 3: MPI/Synergy menus and their functions

Menu	Function
File	Creating, opening, closing, importing, exporting, printing, setting system preferences and saving of projects and models.
Edit	Copying, cutting, pasting, selecting and editing objects, images and properties.
View	Displaying/hiding the various windows, toolbars, etc. within MPI/Synergy. Functions for displaying layers and locking windows and results.
Modeling	Creating, duplicating or querying nodes, curves and regions, creating runners, cooling lines, inserts or mold boundary with the aid of Wizards, defining and activating a local coordinate system or modeling plane, duplicating cavities and diagnosing surface problems.
Mesh	Creating, diagnosing, and fixing meshes. Local mesh refinement is also in this menu.
Analysis	Setting the molding process and analysis sequence, selecting the material, configuring the process settings and all analysis prerequisites necessary for the selected process, including Dynamic Feed Control locations. The Job Manager functions are in this menu.
Results	Setting display options for results, querying results and outputting results to a file. The preferences for the plots properties can be modified from this menu.

Table 3: MPI/Synergy menus and their functions

Menu	Function
Report	Creating, editing and viewing reports based on one or more analyses.
Tools	Creating, importing and editing databases for thermoplastic materials and other materials used in an analysis. The basic commands for the Application Programming Interface (API) are listed here as well.
Windows	Controlling the sub-windows in the display window.
Help	Accessing the Online Help and other information about MPI and Moldflow, including keyboard shortcuts, tutorials and connecting to Moldflow on the Web. Information about the program release and build number are also available from this menu.

Sometimes, menu options may be grayed out and unavailable for use. Which options are available depends on the object that is selected in MPI/Synergy.

Panels

Creating, editing and validating geometry and mesh require substantial interaction with the part model. The panel layout provides an uninterrupted access to the part model in the graphics window can greatly improve user efficiency and productivity, as shown in Figure 31. The Project panel displays either the **Tasks** or **Tools** tab.

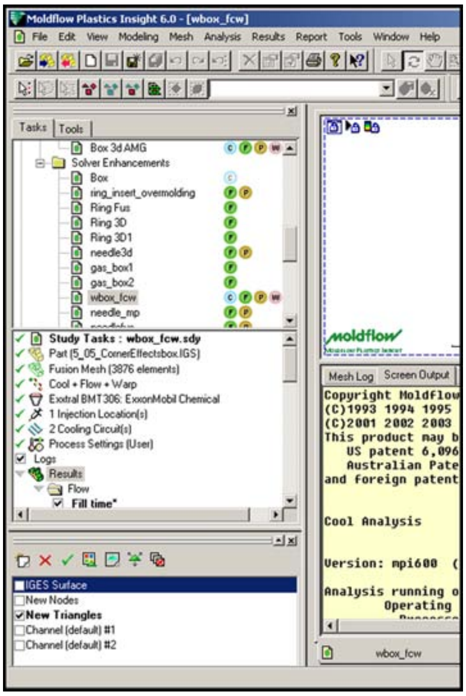


Figure 31: Tasks tab on the Projects panel in Synergy

Tasks tabs

The tasks tab is divided into two sections:

- Project pane.

- Study tasks pane.

A divider bar between the Project pane and the Study tasks pane is movable, so the relative sizes of the two panes can be changed. The Tasks tab is shown in Figure 31.

Project pane

The **Project pane** is the top level of organization in MPI/Synergy. It contains a list of all the studies within a project. A project is equivalent to one "**directory**" on the hard disk, with a collection of MPI analysis files. Within the Project pane, studies can be organized into virtual "**Folders**" and any number of folders can be created to organize studies within one project.

> The study folders created in the Project pane are not "**directories**" on the hard disk. The only exception to this is **report** folders which are in fact created as subdirectories in the project directory.

For each study in the Project pane, you will see one or more icons to the right of the study name. These icons represent the types of analyses that can be run in MPI, and the status of a given analysis.

For example, the icon F indicates the selected analysis is a filling analysis and has not yet been completed because it is not filled. When results are available for the study, the icon is filled in like this F. Several icons can be grouped together to form an analysis sequence such as C F P W, which indicates a Cool, Fill, Pack and Warp analysis (see Figure 32). A Fill and Pack analysis together is called a Flow analysis.

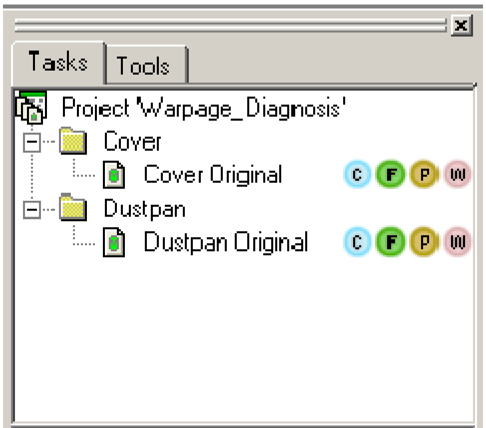


Figure 32: Project pane

Studies

Studies are files that contain all the information about a part to run an analysis. The extension is **.sdy**. The study file contains:

- Imported or created geometry.
- Finite element mesh.
- Analysis sequence.

- Material information.
- Injection location(s).
- Process settings.

Drag-and-drop

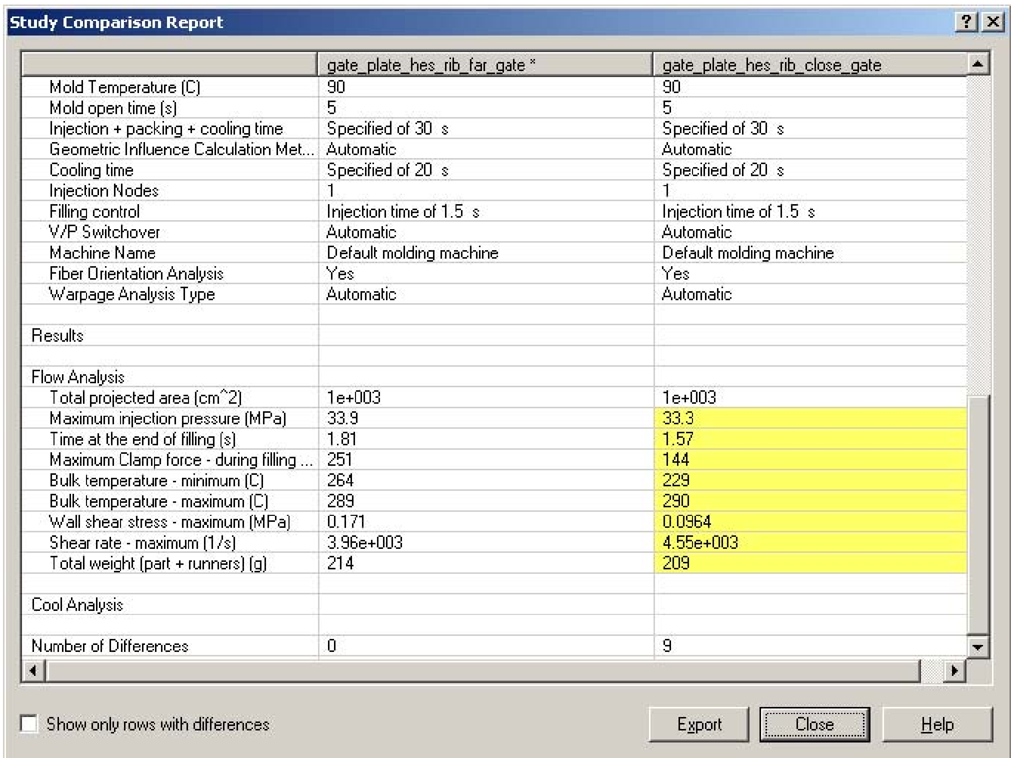
One of the ways you can manipulate the **Project pane** is to drag studies from one place to another. This is useful in many situations.

To open a study, click on the study and drag it into an empty area of the display window. Several studies can be highlighted at one time using a combination of the shift and control keys. Once several studies are selected, drag them to the display window to open them all at the same time.

Studies can be moved from one folder to another in the Project pane by the drag-and-drop method as well.

Compare studies

When two or more studies are highlighted in the Project pane and you click the right mouse button to activate the context menu, one of the options is **Compare Studies.** This tool will bring up a dialog and will show all the highlighted studies and a listing of parameters to compare. The first study in the list is the benchmark that the other studies are compared to. For values that are different, the field will be highlighted in yellow. This table of information can be exported into a comma delimited file. Table 33 shows an example of the compare studies feature.


Study Comparison Report

	gate_plate_hes_rib_far_gate *	gate_plate_hes_rib_close_gate
Mold Temperature (C)	90	90
Mold open time (s)	5	5
Injection + packing + cooling time	Specified of 30 s	Specified of 30 s
Geometric Influence Calculation Met...	Automatic	Automatic
Cooling time	Specified of 20 s	Specified of 20 s
Injection Nodes	1	1
Filling control	Injection time of 1.5 s	Injection time of 1.5 s
V/P Switchover	Automatic	Automatic
Machine Name	Default molding machine	Default molding machine
Fiber Orientation Analysis	Yes	Yes
Warpage Analysis Type	Automatic	Automatic
Results		
Flow Analysis		
Total projected area (cm^2)	1e+003	1e+003
Maximum injection pressure (MPa)	33.9	33.3
Time at the end of filling (s)	1.81	1.57
Maximum Clamp force - during filling ...	251	144
Bulk temperature - minimum (C)	264	229
Bulk temperature - maximum (C)	289	290
Wall shear stress - maximum (MPa)	0.171	0.0964
Shear rate - maximum (1/s)	3.96e+003	4.55e+003
Total weight (part + runners) (g)	214	209
Cool Analysis		
Number of Differences	0	9

Figure 33: Compare Studies

Study tasks list

The Study Tasks pane displays a list of the basic steps necessary to set up and run an analysis. The information shown in the Study Tasks list always relates to the currently active study. The study tasks pane includes the following icons:

Table 4: Study Tasks Icons

Icon	Name	Description
	Translation model	Indicates the original file format of the geometry.
	Midplane model	Indicates the model uses midplane mesh technology.
	Fusion model	Indicates the model uses Fusion mesh technology.
	3D model	Indicates the model uses 3D (tetrahedral) mesh technology.
	Analysis Sequence	Shows the currently selected analysis sequence, for example Fill, Flow, or, Cool + Flow + Warp.
	Material	Shows the currently selected material and provides access to material searching functions.
	Injection nodes	Sets the injection location on the part.
	Circuits	Sets coolant inlet properties and starts the circuit creation wizard.
	Process settings	Sets all the variables for the analyses selected in the analysis sequence.
	Start analysis	Starts the analysis or opens the job manager with a right click.
	Results	Lists all results including screen output, results summary, analysis check, and graphical results.
	Task Done	Indicates the task is completed and information is provided so an analysis can be started.

In addition to all the tasks required to set up the study to the point where an analysis can be run, there is also a listing of all results created once an analysis is completed. An example Study Tasks list is shown below in Figure 34. In this example, all analysis setup tasks have been completed, an analysis has been run, and results are now ready for viewing.

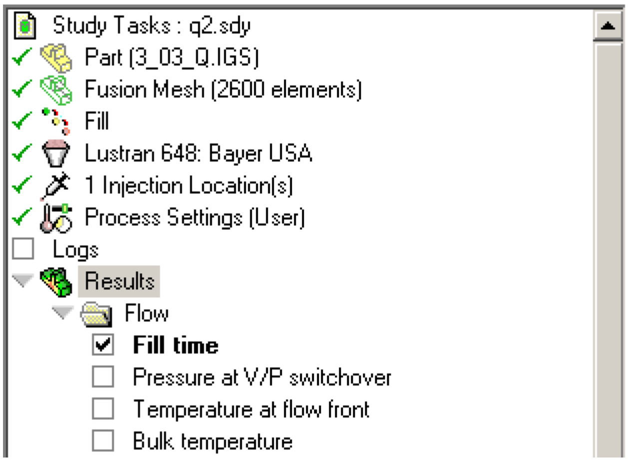


Figure 34: Study tasks list

Tools Tab

Modeling and meshing tools can be found on the newly-created **Tools** tab in Project panels. Now, the part model remains visible in the graphics window without being obscured by tool dialogs, as shown in Figure 35.

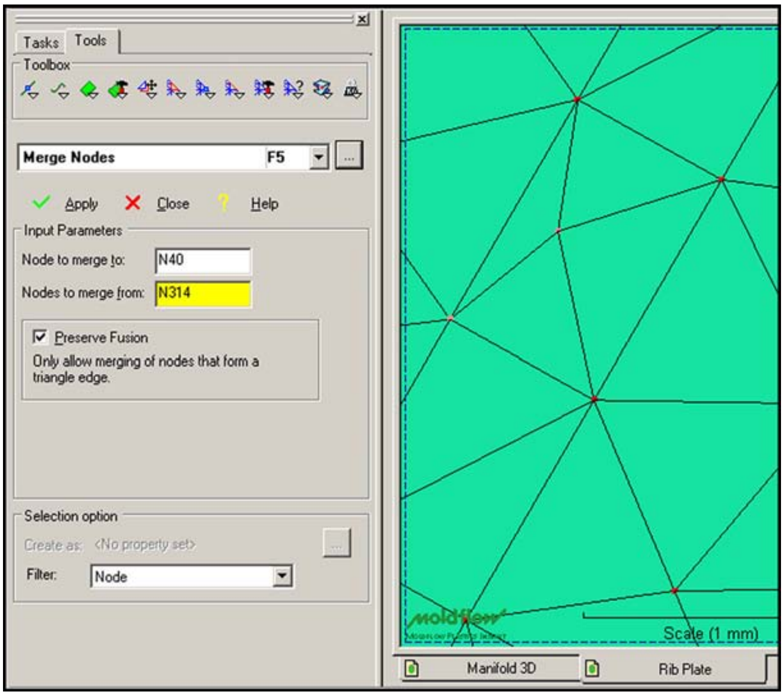


Figure 35: Tools tab in the Project panel

Toolbox

The **Toolbox** is found on the **Tools** tab, as shown in Figure 36. Through the Toolbox, any of the model creation or mesh editing tools can be accessed easily, without having to navigate through the **Modeling** or **Mesh** menus. Every icon on the toolbar has several individual commands under it. When a command is chosen, the combo box below the toolbox is populated with a set of related modeling and mesh commands, and the shortcut key for the current tool is shown. Shortcuts range from F2 to F12. Click on the icon ... to change the shortcut keys for the commands. Table 5 lists the tools in the Toolbox.

Figure 36: Toolbox

Table 5: Toolbox tools

Icon	Tool
	Create Nodes
	Create Curves
	Create Regions
	Surface Tools
	Move/Copy
	Create/Beam/Tri/Tetra
	Nodal Mesh Tools
	Edge Mesh Tools
	Global Mesh Tools
	Mesh Diagnostics
	Set Constraints
	Set Loads

Entity Selection

If a command is active, such as Merge Nodes, and you want to select entities for some other purpose, such as moving entities between layers, the tool must be closed first. Close a tool by pressing the **Close** button on the panel, or by right clicking in the model window and selecting **Finish**.

Layers

Layers is a method of organizing the entities (nodes, elements, regions, etc.) in your model into groups, and then controlling the visibility and display properties of the entities on a per-group basis. The use of layers is very handy when modeling, fixing the mesh or during results visualization. For a model that has been meshed, there will be several default layers including: Default Layer, New Nodes, and New Triangles. There will also be one or two layers for the imported geometry. Layers can also be organized to represent geometries, such as runners, gates, edge, top, side, etc.

> Results are automatically scaled by visible layers. Layers can be created at any time to aid in viewing results.

The Layers pane is by default, located under the Tasks/tools pane. The Layers pane can be a floating dialog. To make the pane float, simple click in the icon area of the pane and drag it of the panel.

> An entity can be only on one layer at a time.

In the layers pane shown in Figure 37 below, several details can be seen including:

- The **bold** layer is the **active layer.** Any new geometry that you create, for example a runner system, will be added to this layer.
- The layers with a tick in the check box ☑ are **visible layers**. All entities assigned to those layers will be visible on the screen.
- The highlighted layer with the blue background Default Layer is the **currently selected layer**. Most of the layer commands work on the highlighted layer. as described in Figure 6.

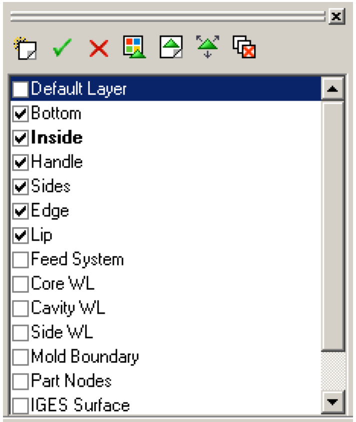
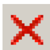
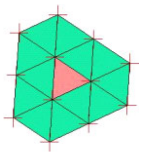


Figure 37: Layers pane

A complete list of commands and actions available for layers are shown in Table 6.

Table 6: Layer icons

Icon	Command	Action
	Create Layer	Creates a new layer. Once created, click on the name to change it.
	Delete Layer	Deletes the highlighted layer. If there are any entities on that layer, you will be prompted to delete the entities or to move the entities to the active layer.
	Activate Layer	Makes the highlighted layer active. The layer name becomes **bold**.
	Layer Display	Allows you to view and edit the color and display methods of the entities on the highlighted layer. You also have the option to hide/ show entity labels, for example element or node numbers.
	Assign Layer	Assigns the currently selected entities in the display window to the highlighted layer.
	Expand Layer	Adds entities to the highlighted layer by a number of levels. If there is only one element on a layer,  and the expand command is used, all of the elements attached to the single element are added to this layer, as well as all of the nodes on the elements. 
	Clean Layers	Deletes all layers with no entities on it.

Layer Display Dialog Components

For a selected layer, the display layer settings can be applied. The layer display dialog is shown in Table 38.

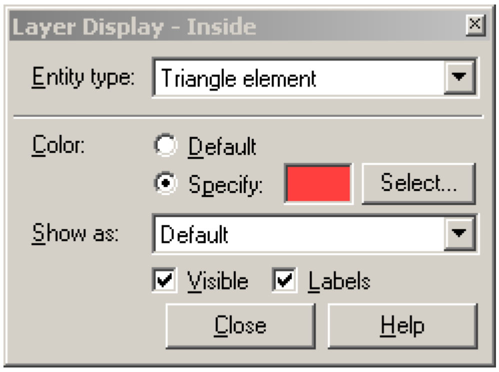


Figure 38: Layer display dialog

The individual components of this dialog and its functions are in Table 7.

Table 7: Layer display commands

Command	Function
Entity type	Select the entity type to be modified from the drop-down list. A layer can have more than one entity type. Each entity can have different display settings.
Color	Specify the display color for the selected entity type within a layer. Click **Default** to use the default color associated with an entity type. Click **Specify** and then **Select...** to choose any other color.
Show as	Specify the display method for each entity type within a layer. From the drop-down list, select a display method, such as: • Default. • Solid. • Solid + Element Edges. • Transparent. • Transparent + Element Edges. • Shrunken.
Visible	Specify whether each entity type within a layer is visible, or hidden. Click in the box to turn visibility on or off.
Labels	Display on or off the labels of each entity type.

Layer context menu

The context menu (right click) for the layers area is particularly useful. In addition to the same commands as the icons on the layer toolbar, there are other commands that are useful. Below in lists the commands on the context menu.

Table 8: Layer commands found on the context menu

Command	Description
Make Active	Makes the highlighted layer active. Described in detail in Table 6.
Assign	Moves selected entities to highlighted layer. Described in detail in Table 6.

Table 8: Layer commands found on the context menu

Command	Description
Display	Changes the display properties for entities on the highlighted layer. Described in detail in Table 6.
Expand	Adds nodes and elements to entities on the highlighted layer. Described in detail in Table 6.
Delete	Deletes the highlighted layer. Described in detail in Table 6.
Rename	Change the name of the highlighted layer
Labels	Turns on entity labels (Node numbers for example) for the highlighted layer.
Show all Layers	Turns on all layers.
Hide all Other Layers	Turns off all layers except the highlighted layer.
Move Up	Moves the highlighted layer up one spot in the layer list.
Move Down	Moves the highlighted layer down one spot in the layer list.

Toolbars

Most of the commands available in the menu are also available on a toolbar. Each toolbar can be moved to any part of MPI/Synergy by clicking on it and dragging. The toolbars can be displayed or hidden using the **View ▪ Toolbars** menu option. Table 9 below describes each toolbar and its function. For a description of any of the buttons on the toolbar, use the **What's This?** online Help feature.

Table 9: Toolbars and their functions

Toolbar	Function
Standard	Opening, saving, printing and editing, and online help.
Viewer	Manipulating the display of the model or results, querying results.
Animation	Controlling the animation of results.
Selection	Selecting entities using a variety of tools.
Analysis	Setting the molding process, analysis sequence, selecting the material, configuring the process settings and all analysis prerequisites necessary for the selected process.
Viewpoint	Selecting standard view rotations, keying-in rotations, and saving views.
Precision view	Setting the view rotation, zoom level and pan position using incremental-change tools.
Modeling	Creating geometry, nodes, curves, or regions manually or using Wizards.
Mesh Manipulation	Creating, diagnosing and repairing a mesh.
Macro	Starting, stopping, and playing a macro
User Macro/ Command Buttons	Assigning macros or commands to 10 user-definable buttons

The buttons on the toolbars are not always available since the functions for the buttons might not apply to the selected object. Any buttons not available are grayed-out. You can create new toolbars, or customize existing toolbars, using the **View** ➡ **Toolbars** ➡ **Customize** menu option.

Creating a customized toolbar like the ones shown in Figure 39 can significantly increase productivity. This is another way to access commands that are in the toolbox quickly. By clicking on one of the command icons in the toolbar, the Tools pane displays the selected tool. As always, the toolbars can be customized to contain only the tools that you use.

Figure 39: Customized toolbars

Context Menu

The context menu is a short menu that pops up after a right-mouse click. The context menu appears with different options, depending on what is highlighted or where the cursor is.

Different context menus appear at the following locations:

- The project name, folder name, or study names in the **Project pane.**
- Every step in the **Study Tasks list.**
- Individual results in the **Study Tasks list.**
- In the **Display Window.**

The context menu often provides a very quick and handy way to access relevant functions of MPI/Synergy. An example of a context menu is shown below in Table 40.

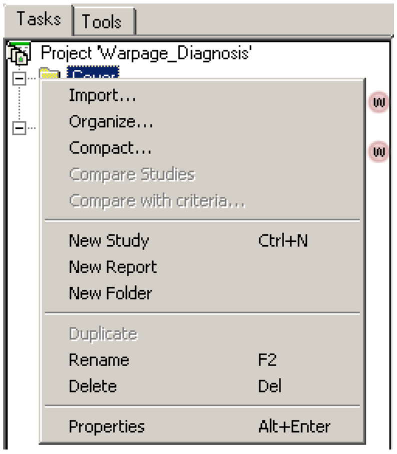
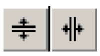


Figure 40: Context menu for the Project pane

Display window

You can have many studies open at one time in the display window. Each open study in the display window is called a document.

Splitting windows

You can split a study document into one, two, or four sections to simultaneously display different results for the same model. An example of a split in two is presented in Figure 41. You can also use the display area to compare the results of several analyses from the same project. Splitting windows can be done two ways, using the command Window ➡ Split, or the icons  found on the Viewer toolbar. When the command is used, the window is split into equally sized windows. To remove the splits, select the border and drag it to an edge.

Locking

There are three ways plots can be “locked”. These include:

- Views.

- ➤ This locks rotating, panning, zooming. Figure 41 has both windows with the view locked.
- Animations.
  - ➤ This locks the animations of the plots so frames in each locked window are displayed at the same time.
- Plots.
  - ➤ This locks the display and plot properties of each locked window. If a result is displayed for one study, they are displayed for all.

Locking can be set or unset by several ways including:

Table 10: Locking and unlocking commands

Icon	Name	Description
	Lock/unlock view	Toggle that locks or unlocks a single window by rotation pan and zoom. Found on the Viewer toolbar.
	Lock/unlock Animation	Toggle that locks or unlocks a single window for animating results. Found on the Viewer toolbar.
	Lock/unlock	Toggle that locks or unlocks a single window for displaying or modifying the properties of a result. Found on the Viewer toolbar.
	Lock all views	Locks the views of all windows that are split within a study and/or all open studies. Found **View** ➡ **Lock** ➡ **All views** command.
	Lock all Animations	Locks the animations of all windows that are split within a study and/or all open studies. Found **View** ➡ **Lock** ➡ **All Animations** command.
	Lock all plots	Locks the plots of all windows that are split within a study and/or all open studies. Found **View** ➡ **Lock** ➡ **All plots** command.
	Unlock all views	Unlocks the views of all windows that are split within a study and/or all open studies. Found **View** ➡ **Unlock** ➡ **All views** command.
	Unlock all Animations	Unlocks the animations of all windows that are split within a study and/or all open studies. Found **View** ➡ **Unlock** ➡ **All Animations** command.
	Unlock all plots	Unlocks the plots of all windows that are split within a study and/or all open studies. Found **View** ➡ **Unlock** ➡ **All plots** command.

The Lock all and unlock all commands are commonly used. Create a custom toolbar for faster access to the commands.

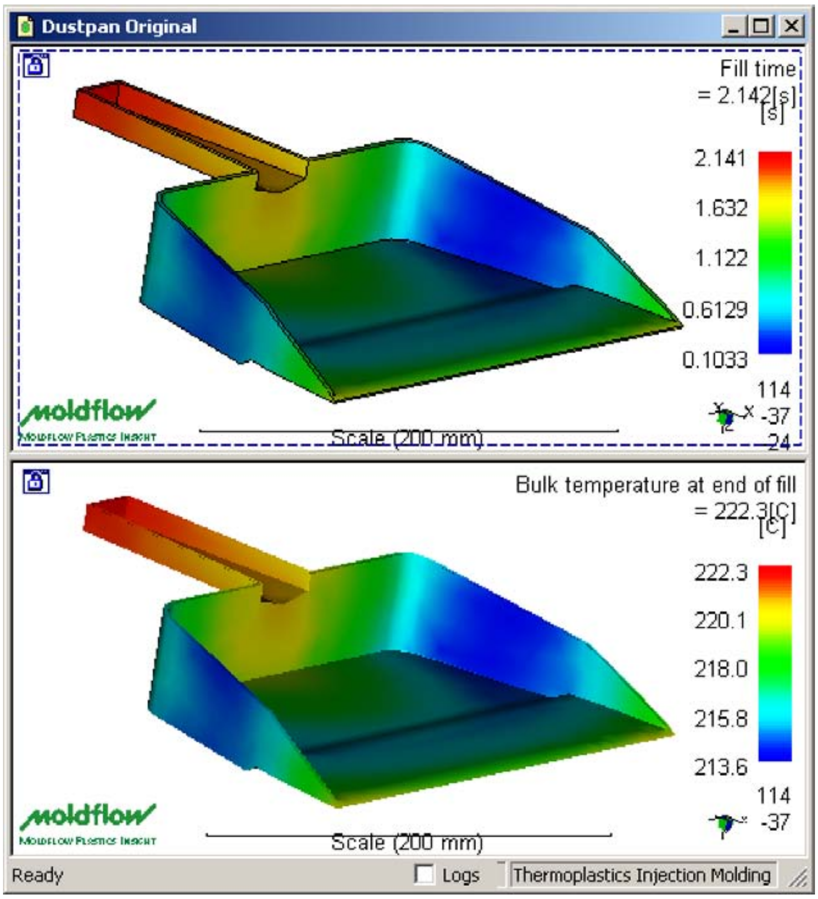


Figure 41: Display window

Wizards

Synergy has several wizards that can be used to automate and significantly speed up the task of geometry creation, to cleanup a mesh, and to set up an analysis. The available Wizards and their function are summarized in Table 11.

Table 11: Overview of MPI/Synergy Wizards

Wizard name	Menu	Use this Wizard to...
Cavity Duplication Wizard	Modeling	Duplicate cavities
Runner System Wizard	Modeling	Model a sprue, runners and gates
Cooling Circuit Wizard	Modeling	Model cooling lines
Mold Surface Wizard	Modeling	Model the mold boundary
Mesh Repair Wizard	Mesh	Diagnose and repair the mesh
Process Settings Wizard	Analysis	Edit the analysis inputs
Plot Customization Wizard	Results	Create custom results
Report Generation Wizard	Report	Create reports

Working in Synergy

Below are a few common tasks required in the process of setting up and running an analysis.

Creating and opening projects

Every time you want to work on a project or run an analysis within MPI/Synergy, a project must be open. If you are starting a new project, a project folder must first be created. If a project exists, the project folder must be opened.

Creating a new project

If you need to start a new project, use the menu command **File** ➨ **New Project** to open the **Create New Project** dialog shown in Figure 42. Specify the folder where you want the new project to be created either by typing in the path in the **Create in** text box, or by clicking **Browse** to locate the folder. This folder will be parent folder for the new project. Now enter a project name. As you type, the project name is automatically appended to the "**Create in**" path. In this example, the project called "**New Project**" will be created in the folder "**e:\My MPI 6.0 Projects\New Project**".

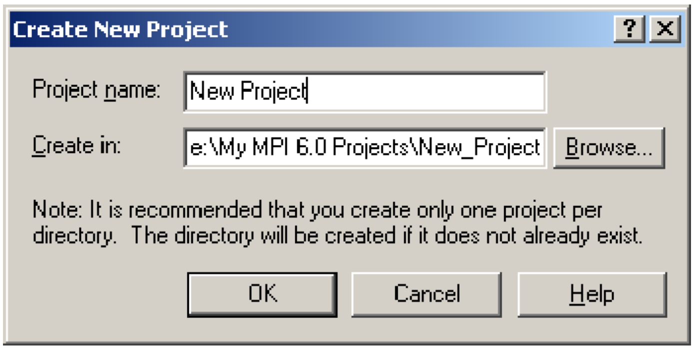


Figure 42: Create New Project dialog

Open an existing project

The procedure for opening a project that already exists is similar to opening a file in a word processor. Use the menu command **File** ➨ **Open Project**, then navigate to the location of the project folder. Click on the ***.mpi** file and click **Open** as shown in Figure 43.

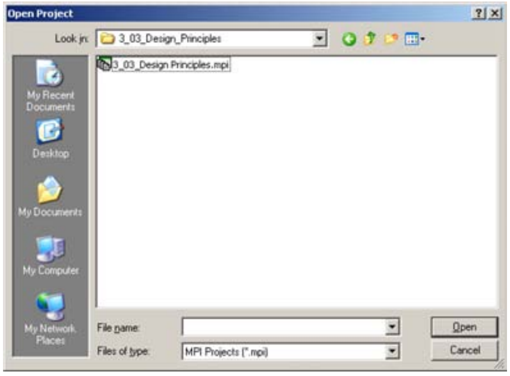


Figure 43: Open Project dialog

Preferences

In the File menu, there is a menu option called **Preferences** that displays a dialog shown in Figure 44, with many settings for Synergy. There are several categories of preference settings, including:

- General
- Mouse
- Results
- MDL (Moldflow Design Link)
- Default Display
- Viewer
- Background
- Help System
- Internet

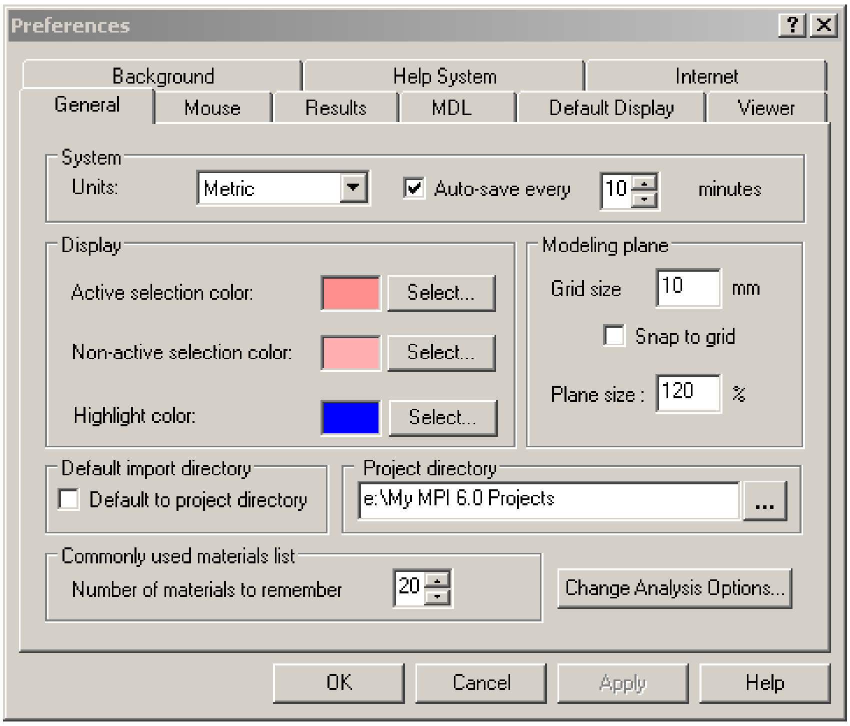


Figure 44: Preferences dialog

There are a few preferences that are commonly changed by users - these are summarized in Table 12 below.

Table 12: Common changes made to preferences

Tab	Preference	Use this preference to...
General	Units	Switch between English and Metric units. This can be done at any time.
General	Auto-save every	Turn the auto-save function on or off, and to specify at what the frequency the auto-save is to be performed.
General	Default import directory	Specify your preferred import directory. If selected, the file import dialog will always open the project directory. If not selected, the dialog will open the last directory where you imported a model. It is sometimes handy to toggle this check box depending on where the files to be imported are located.
General	Modeling plane	Specify the size of the grid on the modeling plane, and to turn the snap-to-grid feature on or off.
Mouse	Middle, Right, Initial mode	Assign commonly used commands to the middle and right mouse buttons. A possible setup would be: • Middle - Rotate. • Middle+Shift - Center. • Middle+Ctrl - Dynamic Zoom. • Right - Pan. • Right + Ctrl - Mouse Apply. • Wheel - Dynamic zoom • Wheel + Shift - Pan X • Wheel + Ctrl - Pan Y The "**Initial mode for new windows**" is the command for the left button. A common setting for this option is **Select**. With the above settings, the most common model manipulation commands can be done with the mouse, saving considerable time.

Table 12: Common changes made to preferences

Tab	Preference	Use this preference to...
Default Display	Various	Set how the various entity types in the model are to be displayed: • Triangle element. • Beam element. • Tetrahedral element. • Node. • Surface. • Region. • STL facet. • Curve. For elements, the display styles available include; • Solid. • Solid + element edges. • Transparent. • Transparent + element edge. • Shrunken. These display settings affect all currently open studies, and all studies subsequently opened. The display options can also be set on a per-layer basis via the Display button on the Layers pane. Layer-based settings override the global settings in the Preferences dialog.
Viewer	Lighting	Set the level of shading in the model display window. By default, the Light shading is set to a maximum, (slider all the way to the right). This may not be the best setting for mesh editing. The maximum setting will create a dark shadow for elements when viewing them at a low angle. Setting the lighting to about 1/3 maximum is normally more acceptable. Results are often viewed best at higher values.
Results	Default results	Set the default results to show in the Study Tasks pane when an analysis is complete. To add more results to the default results list, click Add/Remove, then select them in the All Results list and click >> to move them to the Default results list. There are also options for overcoming memory limitations when viewing results for large models.

Entity selection methods

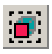
There are several methods of selecting model entities in order to manipulate them in some way. The most common way is to use the Select icon , found on the viewer toolbar. This tool is often used in combination with other tools. Selecting entities is done for a variety of reasons including:

- Select an entity to fix a mesh problem.

- Assign the entity to a different layer.
- Assign properties to elements.

The methods for entity selection include:

Table 13: Methods of selection

Icon	Description
	Click on an element.
	Click and drag the mouse to band-select multiple entities. This will select all entities through the model that are touching or inside the banded boundary.
	Hold down the control key to select multiple entities or banded regions.
	Band-select facing items only.
	Band-select enclosed items only.
	Hold down the shift key to de-select by banding.
	Band-select a circular area
	Band-select a polygon of the users definition.
	Select entities by their properties. This can also be accessed by **CTRL+B**.

The selection methods are available in the **Edit** menu or on the **Selection** toolbar. Selected elements are shown in a color defined by the **Active selection color** option in the **General** page of the **Preferences**. The default color is pink, see Figure 45.

Several selection methods can be used at the same time. A banded selection can be used for making a selection. Then by holding down the control key, the banded selection can be used again. When the control key is depressed, individual elements can be selected by clicking on them.

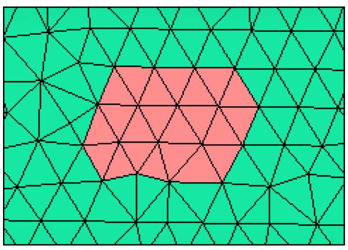


Figure 45: Selected elements

Entity selections can also be saved as a list in the **Select Saved Selection List** on the **Selection** toolbar, shown in Figure 46. When entities are selected on a model, the entire list of entities is updated into the **Select Saved Selection List** field.

Figure 46: Selected entities listed in Select Saved Selection List

To save the list, select the **Save Selection List** icon  and enter a name. To retrieve the saved list for use at a later time, click on the drop-down arrow and select the name that you entered. This is a useful feature if you are performing repetitive tasks with the same group of model entities.

The selection list is integrated with the modeling tools. Notice that the saved selected entities are available for selection in other dialogs as shown in Figure 47.

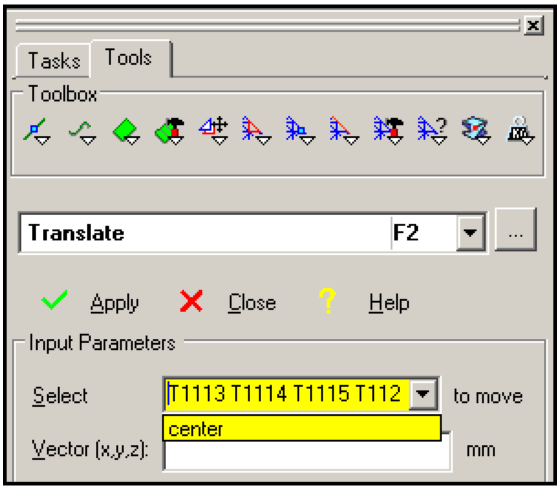


Figure 47: Selection list usage

Working with properties

Properties are characteristics of entities in a model, typically elements. They are grouped together depending on the type of entity. For instance, there are different properties for beam elements that are used to define runners, gates, sprues, water channels etc. There are also properties for triangular elements that are used to define midplane or Fusion models.

When working with properties, there are two basic approaches: edit an existing property or create a new one. The preferred method to use depends on the situation.

Editing properties

To edit an existing property, select one element that has a property you want to edit. Click the right mouse button and select **Properties** at the bottom of the context menu. This can also be done with the menu command **Edit·Properties**. Notice a check box called **"Apply to all entities that share this property"** as shown in Figure 48. This will highlight all elements on the model with the same property. Edit the properties on the various tabs as required, then click **OK**. All the elements will be updated with the new model property.

> Uncheck the "**Apply to all entities that share this property**" if you would like to change the properties of just the highlighted entities.

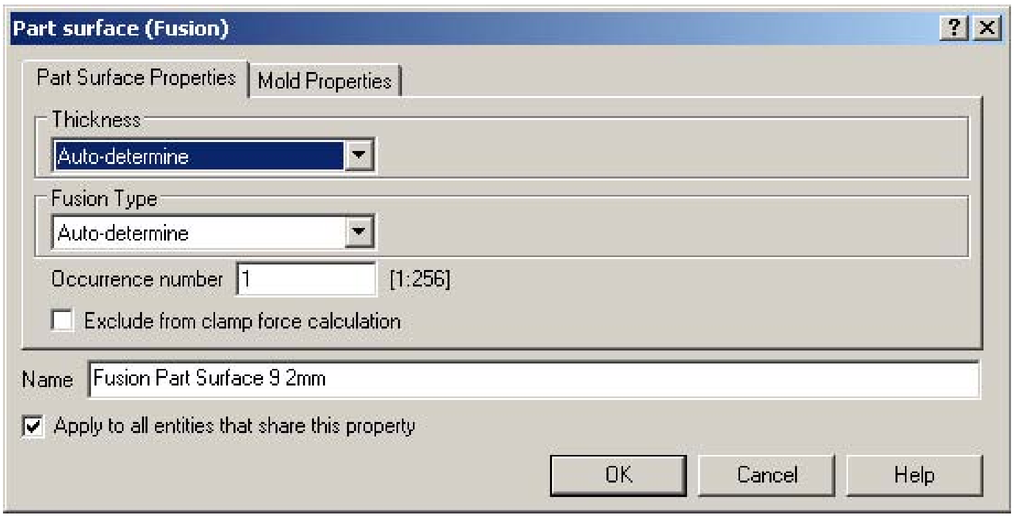


Figure 48: Part surface properties dialog

Creating a property

Sometimes, editing a property will not accomplish the desired outcome. In this case, you may want to create a new property and assign it to specific elements. For instance, you may want to change the thickness of some elements to create a flow leader.

To create a new property:

1. Select the group of elements for which you want to create a property.
2. Select the menu command **Edit ➧ Assign Property**.
3. Click **New** in the Assign property dialog, as shown in Table 49.
4. Select the type of property that you want to create.
    - The actual entries displayed in the list will dependent on the entity type to which you are apply a new property, and the mesh type (Midplane, Fusion or 3D).
5. Enter the property values into the dialog as required.
6. Enter a unique name for the property in the **Name** text box.
    - This makes it easy for you to distinguish properties that you created from those created automatically by Synergy.

7. Click **OK** twice.

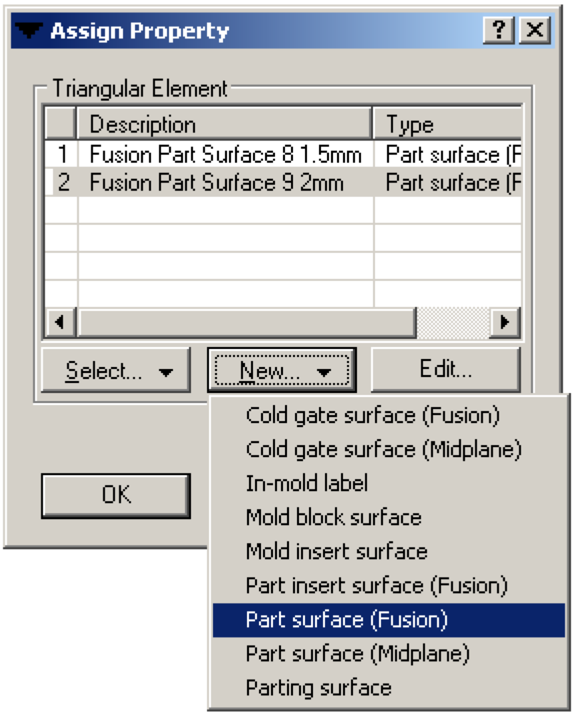


Figure 49: Assign Property dialog

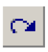
Model manipulation

The tools that are commonly used to manipulate the model are summarized in Table 14. These tools are located on the **Viewer, Viewpoint, Precision view** and **Standard** toolbars, as shown in Figure 50. They are also available in the context menu that appears when you right-click in the display window.

Table 14: Popular tools and their descriptions

Tool	Name	Description
	Rotate	Dynamically rotates the part.
	Pan	Moves the part within the display.
	Banding zoom	Zooms in an area that you select by banding.
	Dynamic zoom	Increases/decreases the model magnification as you hold the left mouse button and drag the cursor up or down the screen.
	Center	Moves the location that you click on to center of the screen. This will become the new center of rotation.
	Measurements	Measures the distance between two nodes or any other location, on the model. The part can be manipulated between picks.

Table 14: Popular tools and their descriptions

Tool	Name	Description
	Fit to Window	Re-sizes the model so the whole part fills the screen.
	View	Series of icons to rotate the model to preset rotations. There is also a text box to enter any arbitrary rotation.
	Incremental pan	Series of icons for panning the model in a particular direction by a small increment.
	Incremental zoom	Series of icons for rotating the model about a particular axis by a small increment.
	Incremental rotate	Series of icons for rotating the model about a particular axis by a small increment.
	Cutting plane	Opens a dialog to define and activate a cutting plane to see inside the model.
	Move cutting plane	Open a dialog to move the active cutting plane. It can be moved manually or animated by entering an incremental value.
	Query results	Click on a result or diagnostic to create a label indicating the value at the location of the click. Hold the CTRL key down to keep the label displayed when multiple locations are selected.
	Query Entity	This does not have an icon by default. The command is Modeling ➡ Query entities. Select one entity and it will display information about the entity information includes: • Triangular element - Nodes, Layer, Thickness. • Nodes - Coordinates, Layer. • All other entities, Layer. Select more than one entity to place on an layer. Enter the entity's number as well.
	Undo	Reverses the last action done
	Redo	Reverses the last undo action
	Action History	Shows a list of the most recent actions and allows you to reverse or restore them.

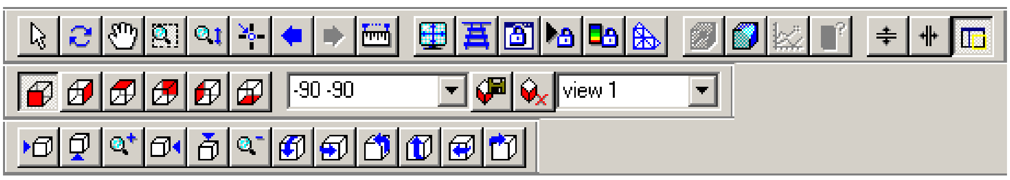


Figure 50: Manipulation tools

What You've Learned

Synergy is the User interface for MPI. It has all the tools necessary to:

- Import a model.
- Mesh the model.
- Clean up the mesh.
- Set up an analysis.
- Review results.

There are many ways that commands can be accessed including:

- Main menu.
- Panel
  - Project View.
  - Study Tasks list.
  - Layers Pane.
- Context menus.
- Toolbars.

Most commands have several methods to activate them.

Synergy provides a comprehensive set of user preferences and customizable toolbars so that users can tailor the program to meet their specific requirements.

CHAPTER 5

How to Use Help

Aim

The aim of this chapter is to learn the many features of the MPI online help system.

Why do it

The MPI online help provides a wealth of information on using MPI, from solver theory to how to interpret analysis results. This chapter will introduce you to the various ways the help can be used.

Overview

In this chapter, you will be introduced and practice with accessing the many help features provided in MPI/Synergy, including:

- Help homepage.
- Help contents.
- Help index.
- Help full-text search.
- Panel/Dialog help.
- Context-sensitive (What's this?) help.
- Favorites.
- Help commands.

Theory and Concepts - How to use Help

Accessing help

MPI's help can be accessed in several ways depending on the information you want and what you are doing, including:

- Help menu.
- Help icons on toolbars.
- Help buttons on panels or dialogs.
- F1 key.

Help Menu

The help menu has a number of ways of getting information as shown in Figure 55 including:

- Search help (main help page).
- What's This?
- Keyboard Shortcuts.
- Tutorials.
- Moldflow on the web.
  - Moldflow Homepage.
  - Moldflow Community Center.
  - Internet Auto Revision Update.
- About.
  - This shows the build of software and other information needed when contacting Moldflow Support.

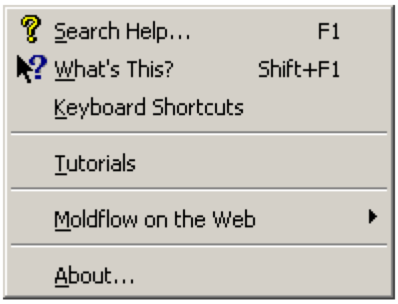


Figure 55: Help menu

Help Icons

On the standard toolbar there are two icons used to access the help. They are:

- The Search help icon  opens the main help interface.
- The What's this icon  allows you to pick on a portion of the graphical interface and get help on this specific topic. A small dialog will open with information about the topic, as shown in Figure 56. This tool will work on individual fields within a dialog or panel.

Figure 56: Help from the What's this tool.

Help buttons on panels or dialogs

On every dialog or panel, there are buttons to get information about the tool. All dialog or panels will have a help button. Dialogs will also have its own What's this button. Figure 57 shows an example of the help buttons on the Generate mesh dialog.

Figure 57: Help buttons on a dialog

F1 key

The F1 key is a context sensitive help. It is normally used to display the help of a result that is displayed on the screen.

Help home page

The help homepage as shown in Figure 58, has two sections, the navigation section on the left and the information display on the right. The help dialog automatically tiles with Synergy so when the navigation page is hidden, Synergy and the help do not overlap. This allows you to keep the help open so you can refer to it while using synergy.

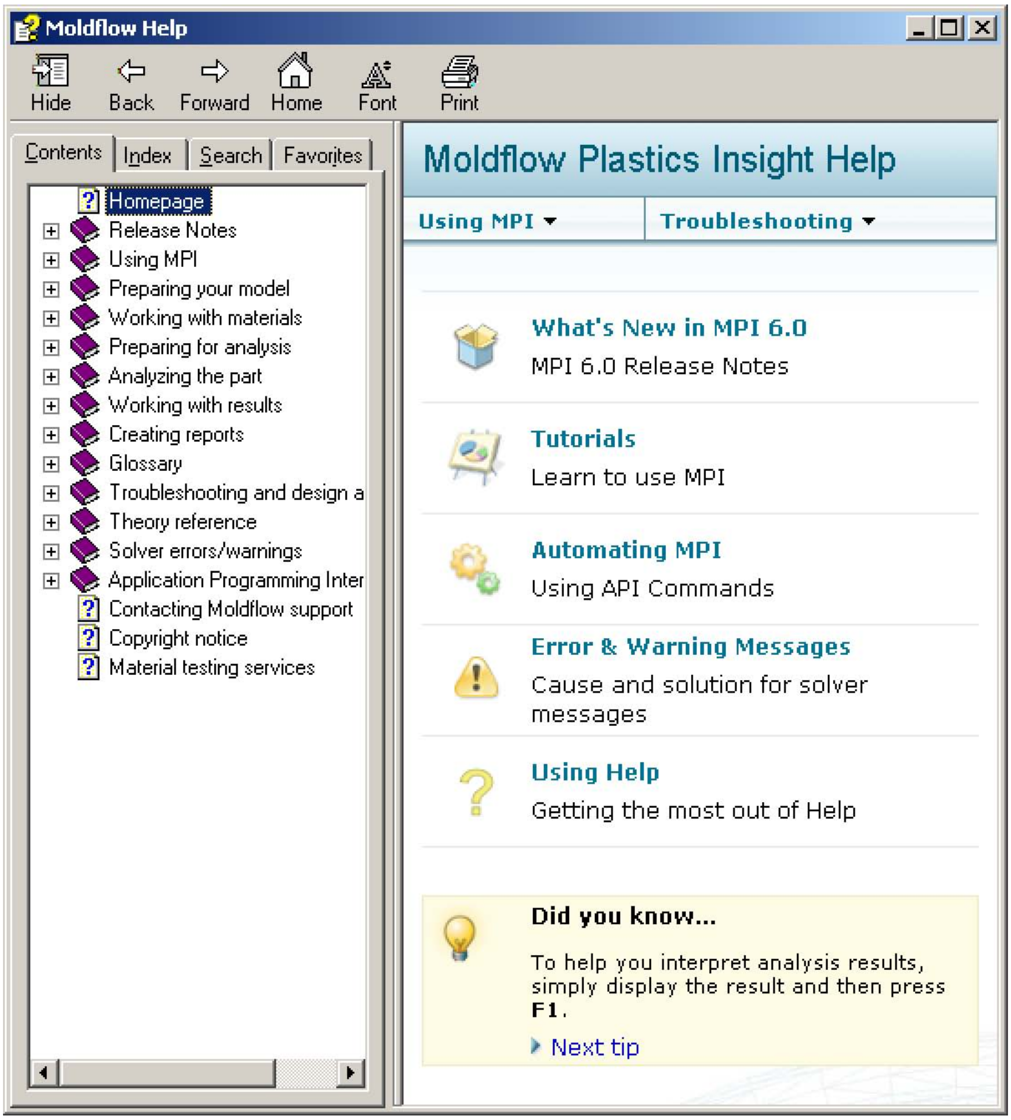


Figure 58: Help homepage

The home page has links to frequently used information very quickly including:

- What's new in the current release.
- Tutorials.
- Automating MPI.
- Errors & Warning messages.
- Using help.
- Tips.
- Using MPI menu.
- Troubleshooting menu.

Using MPI menu

The Using MPI menu has links to many procedural topics in the category of:

- Getting Started.
- Analyzing the part.
- Design Advice.
- Results.

Troubleshooting menu

The troubleshooting menu has information that help you solve problems including:

- Hesitation.
- High volumetric shrinkage.
- Overpacking.
- Racetrack effect.
- Unbalanced flow.
- Air trap.
- Brittleness.
- Burn marks.
- Cracking.
- Delamination.
- Dimensional variation.
- Discoloration.
- Excessive part weight.
- Fish eyes.
- Flashing.
- Flow marks.
- Jetting.
- Short shot.
- Sink mark. and voids.

Error & Warning Messages

When a solver generates an error the solver stops, and when a warning message is generated the solver continues. In both cases, the error or warning code is listed in the screen output file, along with a brief description. If you need more information on the message, click on the **Error & Warnings Messages** link on the home page. The code number can be entered in a search field and the warning or error will be displayed. Figure 59 shows the search field, and Figure 60 shows and example of the results of the search.

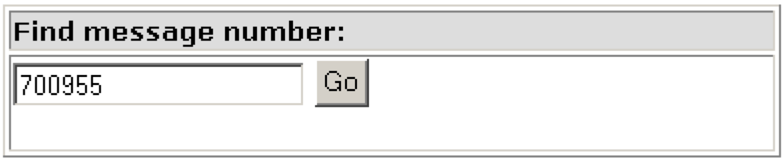
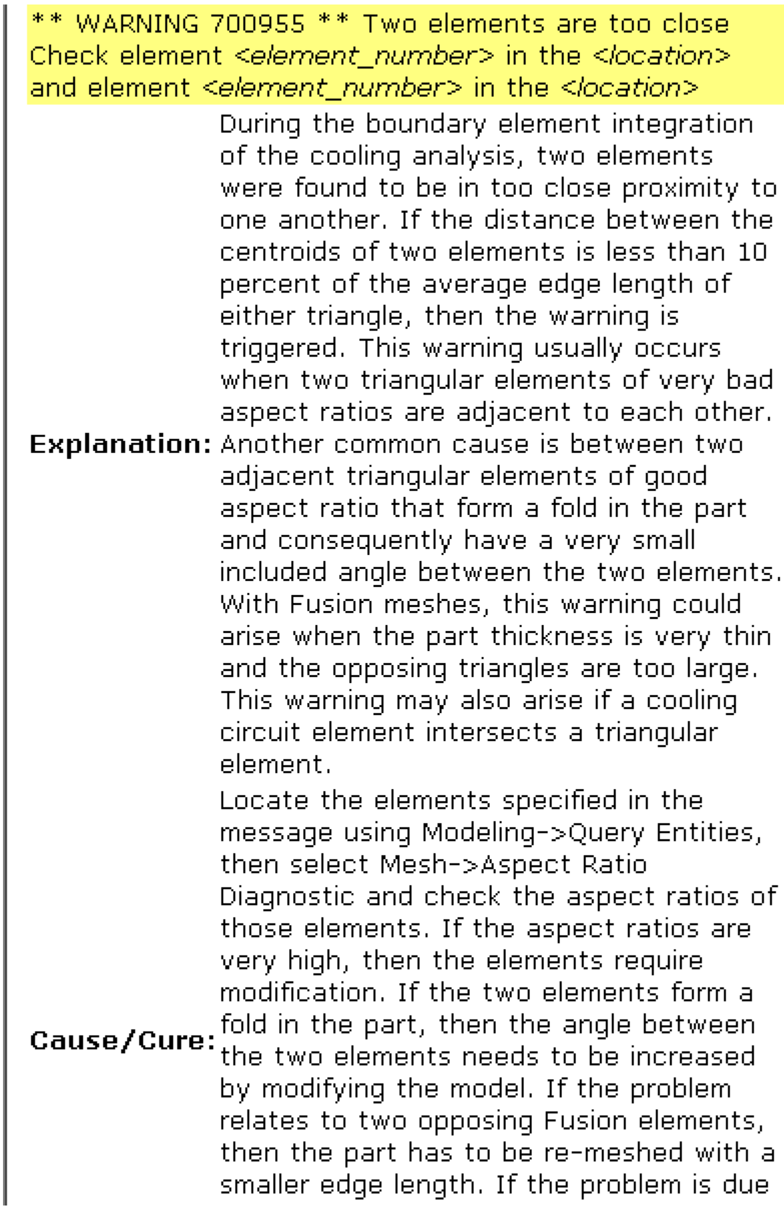


Figure 59: Help error search


** WARNING 700955 ** Two elements are too close
Check element *<element_number>* in the *<location>*
and element *<element_number>* in the *<location>*

Explanation: During the boundary element integration of the cooling analysis, two elements were found to be in too close proximity to one another. If the distance between the centroids of two elements is less than 10 percent of the average edge length of either triangle, then the warning is triggered. This warning usually occurs when two triangular elements of very bad aspect ratios are adjacent to each other. Another common cause is between two adjacent triangular elements of good aspect ratio that form a fold in the part and consequently have a very small included angle between the two elements. With Fusion meshes, this warning could arise when the part thickness is very thin and the opposing triangles are too large. This warning may also arise if a cooling circuit element intersects a triangular element.

Cause/Cure: Locate the elements specified in the message using Modeling->Query Entities, then select Mesh->Aspect Ratio Diagnostic and check the aspect ratios of those elements. If the aspect ratios are very high, then the elements require modification. If the two elements form a fold in the part, then the angle between the two elements needs to be increased by modifying the model. If the problem relates to two opposing Fusion elements, then the part has to be re-meshed with a smaller edge length. If the problem is due

Figure 60: Help warning explanation

Using help

The link on the home page called Using help provides detailed information about how to use the help system. In particular, how to search using boolean operations.

Help contents

On the navigation section of the help there is a Contents tab. Click it to display the help organized as a table of contents. This organizes the help according to topic. Figure 61 shows the contents at its first level some topics are several levels deep. This is a good way to get an overview of the topics covered in the help. Using the Contents is particularly efficient when looking at the Theory reference, and the Glossary.

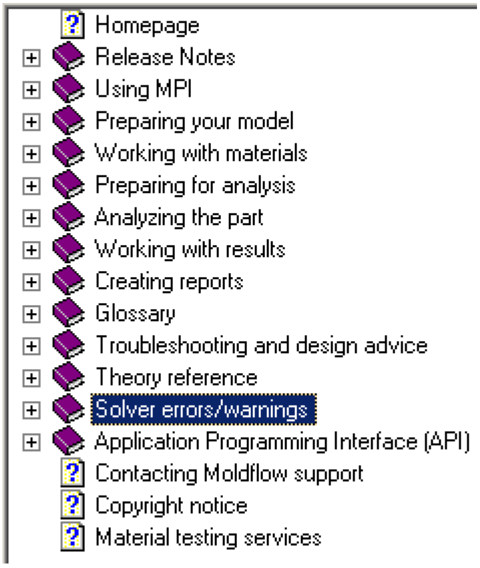


Figure 61: Contents page of the help

Help index

The index tab on the navigation side of the help displays the index for the help. Like a book index, keywords are listed in the index. Enter a keyword or words and the index will go to that part of the index. Figure 62 shows an example of using the help. As letters are typed, the location in the index is changing. Click on an entry to see the help.

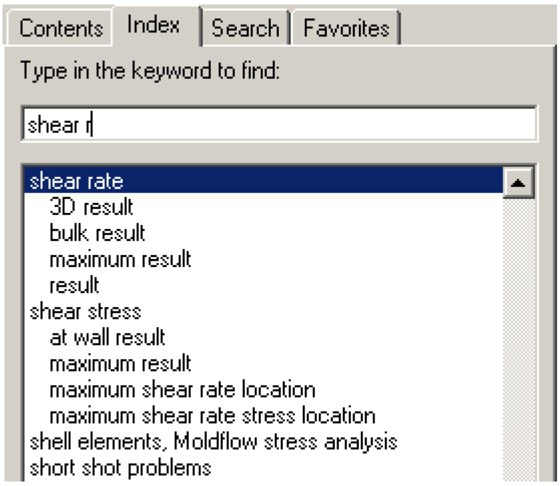


Figure 62: Index page of the help

Help full-text search

The search tab allows you to search all the text in the help topics. To aid in the searching, several operators are used with the search including:

- AND (default)
  - Searching for **injection and molding** lists topics that contain both keywords.
  - Removing the **and** will result in the same output.
- OR
  - Searching for **injection or molding** lists topics that contain either or both keywords.
- "Phrase"
  - Searching for **injection molding** lists topics that contains the exact phrase.
- NOT
  - Searching for **injection not molding** lists topics that contains injection but does not contain molding.
- NEAR
  - Searching for **injection and molding** lists topics that contain both keywords in close proximity to each other.

Favorites

The favorites tab allows you to add the currently displayed page to a list of bookmarked pages. The title of the topic can be set. Click the Add button to create the bookmark. Double click the bookmark to go to the page or highlight it then hit the display button. Highlight the bookmark and hit the remove button to delete the bookmark.

Help commands

The toolbar of the help window has the following commands as shown in Figure 63 including:

- Hide - This hides or shows the navigation pane of the help.
- Back - Goes to the previous help topic.
- Forward -Goes to the next help topic when Back has been used.
- Home - Goes to the home page of the help.
- Font - Toggles through a number of different font sizes for the help.
- Print - Prints the current help topic.

Figure 63: Help window commands

What You've Learned

Accessing help is done a number of ways including:

- Help Menu.
- Help Icons.
- Help buttons on panels or dialogs.
- F1 key.

Help's homepage is has a number of links to useful information including:

- Menus with commonly accessed subjects:
  - Using MPI menu.
  - Troubleshooting menu.
- What's new in a release.
- Tutorials.
- Automating MPI.
- Error & Warning Messages.
- Using help.
- Tips.

The help has a navigation pane that includes:

- Help contents.
- Help index.
- Help full-text search, that can be used with the operators:
  - AND (default).
  - OR.
  - "Phrase".
  - NOT.
  - NEAR.
- Favorites

The toolbar for the Help window has the following commands:

- Hide.
- Back.
- Forward.
- Home.
- Font.
- Print.

CHAPTER 6

Quick Cool-Flow-Warp Analysis

There is no theory and concepts for this subject.

CHAPTER 7

Flow Analysis Steps

Aim

To review the steps involved in running a flow analysis.

Why do it

Every part that is analyzed has a different set of objectives. However, the basic procedure for performing the analysis is the same. An understanding of the basic steps is required, so when a specific problem is determined, the steps needed to solve the problem can be identified.

This chapter will look at the procedures needed to get a flow analysis done. How a flow analysis fits in with a cooling and warpage analysis is also examined.

Overview

To determine the analysis steps necessary to complete a project, a clear understanding of the problem or objectives of the analysis are required. This is the most critical step of the process. Once the problem is understood, determining the steps required to solve the problem is an easy task.

This chapter will concentrate on filling related problems. Most issues that are addressed with MPI are solved or significantly influenced by filling of the part. This chapter will also discuss packing and to a lesser degree, cooling and warpage.

Theory and Concepts - Flow Analysis Steps

Moldflow Design Philosophy

Early developments at Moldflow not only included software, it also developed a strong design philosophy for using the software. At the heart of this Moldflow design philosophy is the **Analysis Sequence**. Below there is a discussion of analysis steps that are required to complete the project. The underlying principle behind this is the analysis sequence. The analysis sequence says you should conduct the analysis in the following order:

1. Determine the number of gates.
   - This is primarily driven by pressure requirements.
   - The pressure required to fill the part should be well under the capacity of the machine.
   - A conservative guideline is that the fill pressure for the part should be half the machine pressure. For a typical machine, this is about 70 MPa (10,000 psi).
   - Add gates as necessary to reduce the pressure to fill.
2. Position the gates for balanced filling.
   - The gate position should produce a balanced flow front within the part, with no underflow or over-packing effects.
   - If the filling pattern cannot be balanced by changing the gate position, flow leaders or deflectors can be used to balance the flow.
3. Ensure the flow pattern is unidirectional.
   - The filling pattern should be straight and uniform.
   - In addition, there should be no problems with hesitation, underflow, or weld lines.
4. Design the runner system.
   - The runners should be designed in such a way as to aid in achieving the required flow pattern.
   - Runners may need to be sized to achieve the desired filling pattern on larger multi-gated parts.
   - They should be balanced, and have minimal volume.

The first 3 steps can be regarded as part optimization. They should always be done first. They can't be considered in isolation, they must be considered together. When adding additional gates, consideration must be given to the position of the gates and the effect on balance and fill pattern. Of course, considerations of the tool layout and restrictions on gate locations must be evaluated.

The design of the runner system should complement the part optimization. Runner systems should be designed so that the runners don't limit the cycle time, or control the process. For instance, if the runner diameters are too large, the cooling time of the runners will control the cycle time of the tool. If the runners are not well balanced, it is possible to compensate for the poor balance by controlling the injection time, however the processing window will be narrowed significantly.

Project Design Procedure Using Moldflow

When using Moldflow to optimize the design of a part or to troubleshoot a problem, the following procedure should be followed.

1. Determine the analysis objectives for the project.
    - The most important part of the project is to have a clear idea of the intended outcomes of the analysis. This point can’t be over emphasized.
    - What problems are to be solved or prevented?
    - Why does the analysis need to be done?
    - Without clear objectives for the analysis, the time spent on any analysis is not well spent.
2. Discuss the project with all disciplines involved in the project.
    - There are 4 main groups of people that should be involved with the analysis. They include:
        - Materials.
        - Part design.
        - Mold design/build.
        - Production/processing.
    - Normally, the analyst is in one of these groups. Rarely if ever, however, is the analyst an expert in all of these areas.
    - It is important that the needs, concerns, limitations, etc. of each group be identified and considered when doing the analysis.
3. Utilize your previous experience when working on the project.
    - The analyst must draw on previous experiences from other analysis projects, and his/her general engineering knowledge, when working on a project.
4. Apply the Moldflow Design Principles.
    - When working on the project, it is important that the analyst apply the Moldflow Design Principles to prevent and/or minimize problems with the part caused by not taking into account polymer flow behavior.
5. Apply the Moldflow Design Rules.
    - Apply design rules when interpreting the results to avoid or minimize such problems as excessive temperature loss during filling, or shear stress and shear rate limits for the material being exceeded.

6. Interpret results and make changes where necessary.
    - Using MPI to optimize the design of a part is an iterative process.
    - Make changes based on careful interpretation of analyses already run.
7. Discuss changes with all disciplines involved in the project.
    - When changes are made, make sure they are practical and acceptable to everyone involved in a project.
    - For example, one solution for solving a problem may be acceptable for the tooling group, but it may not be practical or possible from the point of view of production.
8. Repeat analyses until acceptable results are achieved.
    - Use MPI as an iterative tool. Many iterations may be required to come up with a practical solution.

Optimize Fill

Filling is the most important phase of the injection molding process. If filling is not done correctly, it is impossible to correct problems caused by filling issues in packing or cooling. The basic procedures for doing a filling analysis are similar, even with the wide variety of objectives. A typical analysis may include the following steps:

1. Determine the analysis objectives.
2. Prepare the finite element model.
3. Select the material.
4. Select the gate location.
5. Select the molding machine.
6. Determine the molding conditions.
7. Set the molding parameters.
8. Run the analysis.
9. Review the results.
10. Solve filling problems.

The steps listed above are shown in Figure 80. Several of these main steps have details sub steps including:

- Prepare the finite element model
- Select the gate location
- Determine the molding conditions

Actually solving the filling issues identified can require many iterations going all the way back to the CAD system because the part needs to be re-designed. The list of problems in Figure 80 is an abbreviated list of the problems that can be solved with a filling analysis. The steps are described in more detail below.

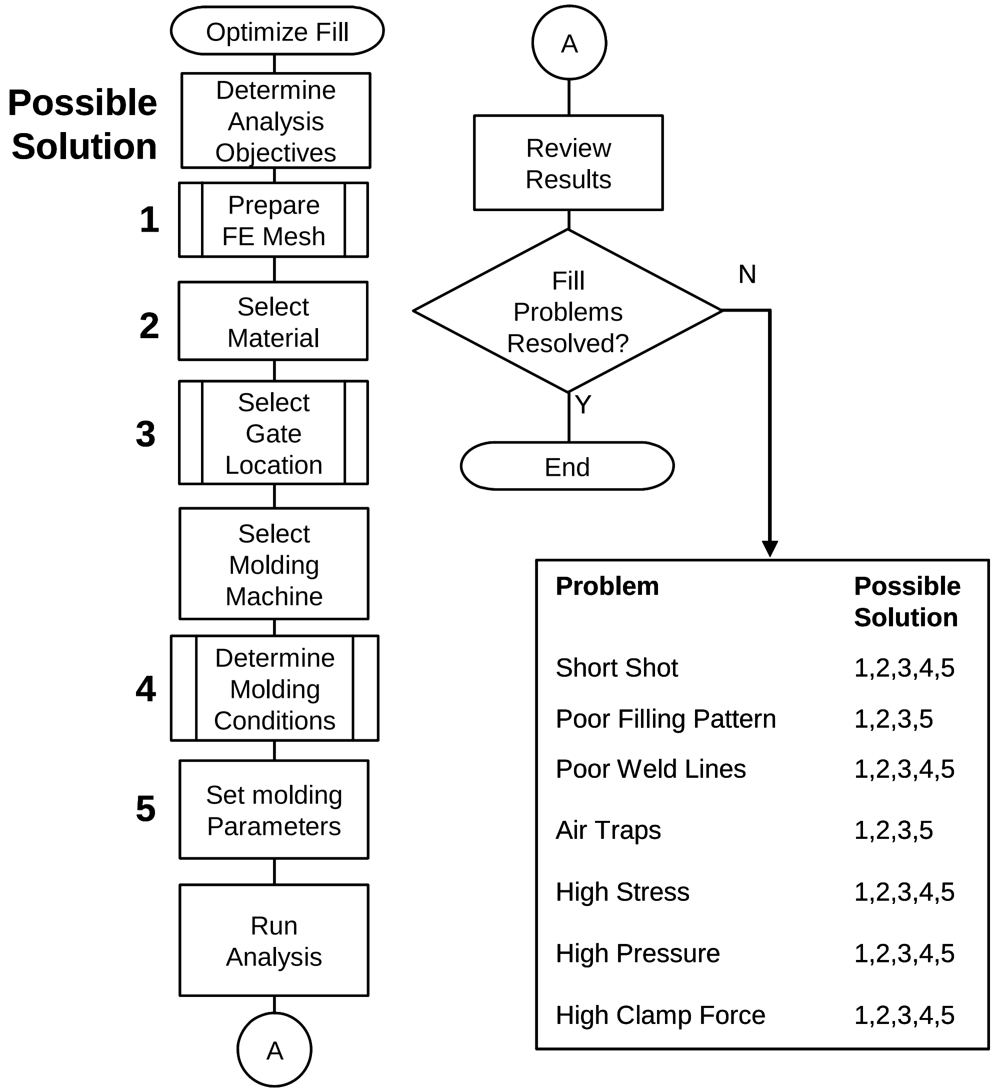


Figure 80: Steps required to optimize the filling of a part

Determine Analysis Objectives

Every part that is analyzed has a different set of constraints in the form of objectives, restrictions, and guidelines. These constraints must be taken into consideration when doing an analysis.

The objectives defined for a part are as varied as the parts that can be injection molded. However, below is a list of analysis objectives that you could have. Some are just flow analysis related; others will require cooling and warpage analysis. Possible objectives include:

- Will the part fill?
- What material will work best for my part with regards to fill properties, i.e. pressure, shear stress, temperature distribution, etc.
- What processing conditions should be used to mold this part?
- Where should the gate be located?

- How many gates are required?
- Where will the weld lines be, and will they be of high quality?
- Will there be any air traps?
- How thick can the part be made?
- Is the flow balanced within the part with the fixed gate location?
- Are ribs too thin to fill completely?
- Are ribs so thick that they shrink too much?
- Can the part be packed out well enough?
- Will this snap fit break during use?
- Can the part be filled and packed in the press specified for the job?
- Are the runners balanced?
- What size do the runners need to be to balance the fill?
- Is the runner volume as small as it can be?
- Is the gate too big or too small?

This is not a comprehensive list, but it gives you an idea of what can be done

Prepare FE Mesh

The sub step of preparing the finite element mesh is flow charted in Figure 81. The steps include:

- Prepare the CAD model.
- Import the CAD model.
- Set the mesh density. This can be done both locally and globally.
- Generate the mesh.
- Evaluate the mesh.
- Determine if the initial mesh is worth using or should the part be remeshed.
- Clean up the mesh.
- Convert to 3D or midplane if necessary.

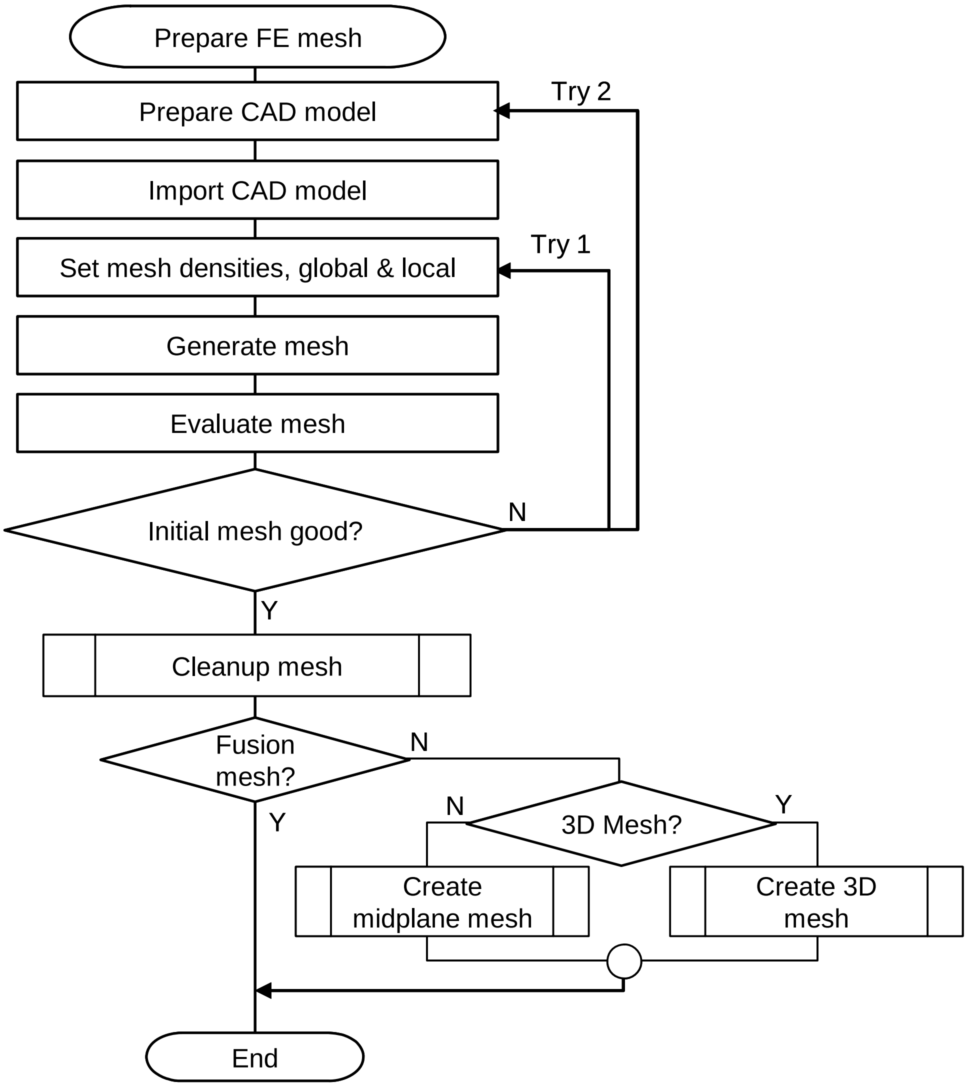


Figure 81: Steps required to prepare a finite element model for analysis

Prepare the CAD model

For most CAD systems, the model needs to be cut in a format that is readable by Synergy.

Import the Cad model

This is simply a file read function. It is just like opening a document in a word processor. When you read in a format like STL or IGES, the geometry is imported but it can't be used for analysis until it is meshed. In the case of mesh formats like Patran, you may need to cleanup the mesh after importing.

Set the mesh density, global and local

The global mesh density setting defines the nominal edge length for elements. The smaller the global edge length, the larger the total number of elements on the model. The global setting is the single most important setting to get an acceptable mesh on the part. When NURBS surfaces are imported, there are several other options available to help control the mesh density including local mesh densities. This is typically done to ensure that small details in the model are properly represented, while meshing most of the part with a lower mesh density to optimize the total number of elements.

STL models are imported as a single entity, so you cannot set local mesh densities immediately after importing. MPI does however provide a tool to create regions from the STL model - local mesh densities can then be assigned to those regions.

Generate the mesh

Once the mesh densities and other options are set generating the mesh is just clicking a button. The time required to mesh the part will depend on the size and complexity of the part.

Evaluate the mesh

After creating an initial mesh, you need to inspect it to decide whether the quality of the mesh is acceptable. If the mesh is not acceptable, the mesh density is too fine or too course for the part as a whole. You should then remesh the part with a different global edge length value. Sometimes the geometry in the CAD system should be changed to get cleaner geometry read into Moldflow. The product Moldflow CAD Doctor may help in preparing the geometry for import.

Clean up the mesh

After meshing with a suitable mesh density, you may need to clean up the mesh. Normally some cleanup is necessary if the mesh was translated in from another system. MPI provides a mesh statistics report to help you check the quality of the mesh, and a series of diagnostic displays to locate and highlight specific problems. To help you clean up the mesh, there is a Mesh Repair Wizard, and mesh cleanup tools. The Mesh Repair Wizard is an automated tool that will find and fix most mesh problems. The mesh cleanup tools include 20 tools for cleaning up the mesh. Some of these tools are fully automatic, such as Auto repair and Fix Aspect Ratio, the others, however, are manual tools for fixing specific problems.

Convert to 3D or midplane if necessary

If the mesh is not Fusion but 3D or midplane, additional steps are necessary. These would include converting the mesh and ensuring the mesh is clean and ready to use for an analysis.

Select the material

To perform an analysis, you need to select a suitable material in the material database. In most cases, the material to use will be prescribed so you need to locate that material in the database and use it for all analysis work. In other cases, one of the analysis objectives may be to determine a suitable material. There are several techniques available to find a specific grade of material, assess the quality of the material data, find a substitute material, and compare materials within the Select Material dialog.

Select the gate location

The gate location on the part may be fixed, there may be two or three choices, or the optimum gate(s) locations may need to be found as a major part of the analysis process. In each of these cases, you need to select an initial injection location. To decide on the final gate location(s), you may need to run several filling, or even flow (filling + packing), analyses.

Select the molding machine

Most analysis situations do not require information about a very specific make of molding machine, so the default molding machine can be used. There may be cases where specific molding machine information is required. The default molding machine has an injection pressure limit of 180 MPa, (~26,100 psi). This is above the pressure capacity for many molding machines. A good design rule is to ensure the pressure required to fill the tool is at most ~75% of the machine capacity. If the exact machine is not known, use a limit of 100 MPa, or 15,000 psi. You can define a custom molding machine with this design limit, or simply use this value when interpreting the results. If you do define a custom molding machine and the injection pressure limit is reached during the analysis, a warning message is displayed in the screen output and the solver modifies the injection profile to keep within the pressure limit.

The default clamp tonnage limit is set very high, at 7000 tonnes. If the clamp tonnage could be an issue, you should select a specific molding machine. This can be done in one of 3 ways:

- Edit the settings of the default molding machine for the current study. When the study is duplicated, the settings will be part of the new study.
- Select a specific molding machine from the machine database.
- Select a generic molding machine from the machine database and modify the settings as necessary.

A good design rule is to ensure the pressure required to fill the tool is at most ~75% of the machine capacity.

If the exact machine is not known, use a limit of 100 MPa, or 15,000 psi.

Determine the molding conditions

The molding conditions to use in an analysis may be mandated, at least as a starting point. Alternatively, suitable molding conditions can be determined from a Molding Window analysis, and then used, perhaps with slight modification, in subsequent filling analyses. In addition to optimizing the molding conditions, a Molding Window analysis can also be used as a quick initial analysis to compare materials or gate locations. You can save a significant amount of analysis time by determining good processing conditions before running filling analyses.

Set the analysis parameters

Analysis parameters, in addition to the "molding conditions", include the velocity/pressure switch-over point, the packing profile, the cooling time, solver settings, process controller settings, the mold material, and others.

Default values for these analysis parameters are normally acceptable for an initial analysis; however, depending on the problems found in earlier analyses and the objectives of the analysis, these may need to be changed.

Run the analysis

This normally refers to a fill analysis or a flow analysis. A fill analysis stops when the part volume is just filled to 100%. The flow analysis is a fill analysis but continues through the packing and even cooling phases of the molding cycle.

You can identify and resolve a number of molding issues using fill analyses before running a flow analysis.

Review the results

After the analysis is finished, the results are reviewed and compared to the analysis objectives. Normally, problems are found, and/or "what if" questions arise, requiring additional analysis.

Solve filling problems

Once a problem has been identified, solving the filling problems is typically an iterative process requiring several analyses. This iterative process could have the analysis go back all the way back to the CAD system for part re-design, or any other step after that.

To this point, the assumption has been the problem is filling related and not packing, cooling or warpage. Optimization of the filling phase is the first step to optimize problems primarily related to other molding phases.

Optimize Flow

The flow chart for optimizing flow is shown in Figure 82. Optimizing the Flow within the part is just an extension of the filling. Additional steps include:

- Balancing the runners.

- Optimizing the cooling.
    - This is optional but recommended.
- Optimizing the packing profile.

Balance the runners

Once the filling of the part is optimized, the runner system can be analyzed, sized, and balanced. This includes sizing the gate and sprue. Depending on the objectives of the project, runner balancing may not be done. The project could end once the filling is done. However, to do a complete analysis, a proposed runner system should be analyzed.

There could be several iterations to try different gate sizes, runner sizes/configurations within this step.

Determine the packing profile

Once the runner system is sized, packing of the part can be investigated. Although a flow analysis can be done without a gate or runner, it is not recommended if you are interested in how the part is going to be packed out. The freeze time of the gate and runners significantly affects the packing of the part. Without a runner and gate, the packing analysis will be less accurate.

A cooling analysis should also be done before the packing is finalized. Heat transfer dominates the packing process. The filling analysis assumes a constant mold temperature. A cooling analysis determines the mold surface temperature distribution on the part making the packing analysis more accurate if done.

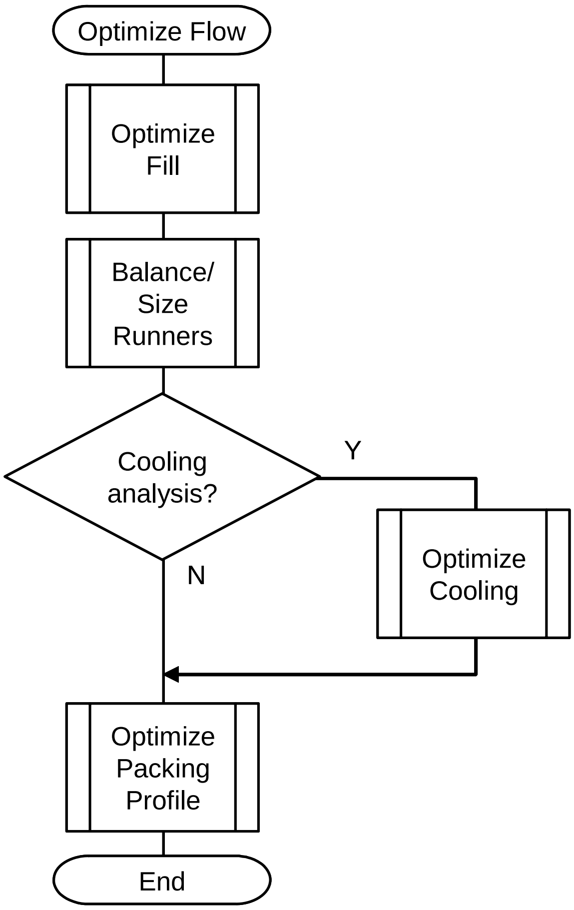


Figure 82: Steps required to optimize the flow (filling + Packing) in the part

Optimize Part

Once the packing is optimized, the optimization of the part's warpage is the last step. This is shown in. Figure 83. Optimization of warpage is a complex process. This is shown in Figure 84.

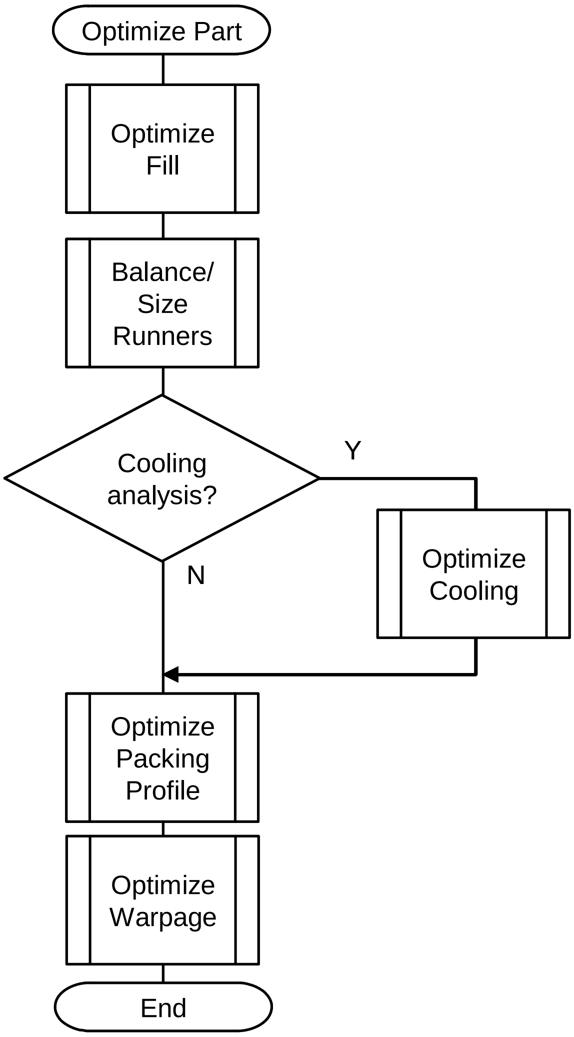


Figure 83: Steps required to optimize the part

Optimize Warpage

To optimize warpage, all the other steps described above must be done. The better job is of optimizing, filling, cooling, and packing, the better the warpage will be. When done right, the warpage analysis is validation that the previous steps are done correctly.

Determine warpage magnitude

The first new step of warpage optimization after the previously described steps is determining the warpage magnitude, as shown in Figure 84. This is the validation step. To evaluate the warpage, there must be an understanding of how the warpage is defined and what tolerances are associated with it. This is often the most difficult part of the process.

Determine the cause of the warpage

If the warpage is out of tolerance, the next step in solving the warpage problem is determining the cause. The cause of warpage is broken down into 4 causes including:

- Differential cooling.
- Differential shrinkage.
- Orientation effects.
  - For Midplane and Fusion this applies to filled and unfilled materials.
  - With 3D, this only applies to fiber filled materials.
- Corner effects.
  - This only applies to Midplane and Fusion.

Reducing the warpage

Once the cause of warpage is known, changes to the part, material or process can be made to solve the warpage problem. This often requires many iterations.

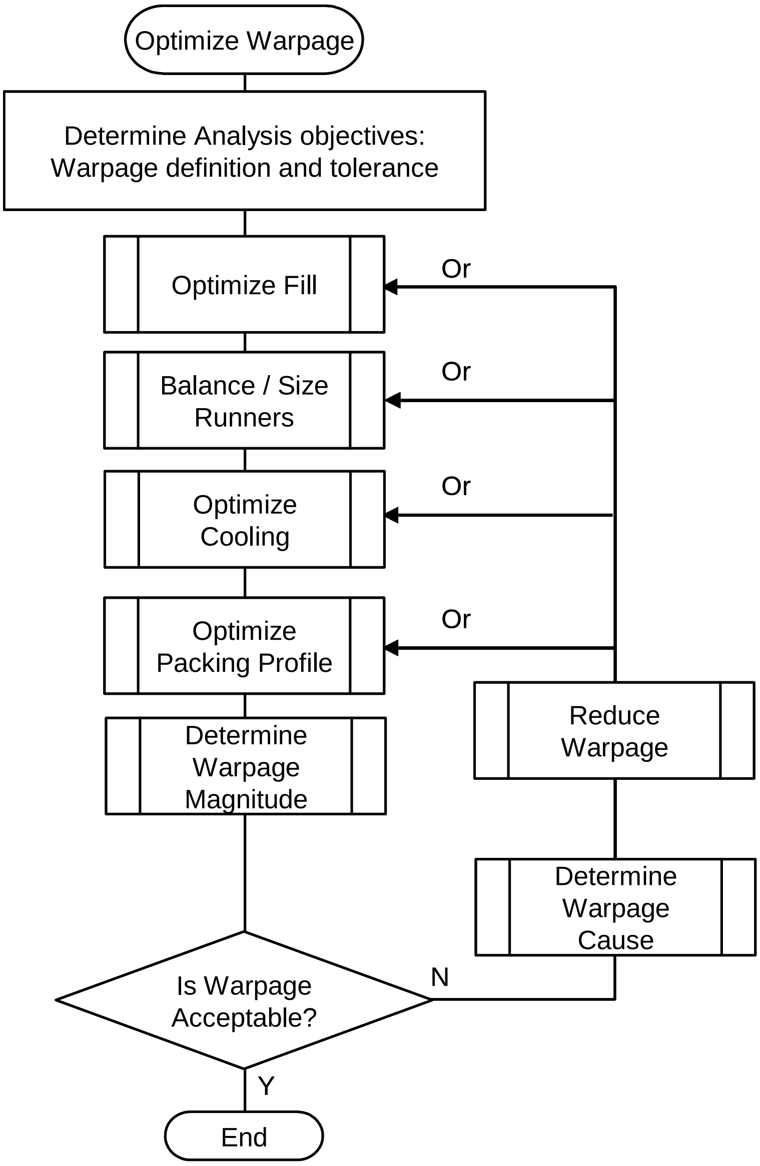


Figure 84: Steps required to optimize a parts warpage

What You’ve Learned

The sequence of steps required to do a flow analysis depends very much on what the objectives of the project are, and what analysis sequences are needed to achieve those objectives. The basic steps for a flow analysis are:

1. Determine the analysis objectives.
2. Prepare the finite element model.
3. Select the material.
4. Select the gate location.
5. Select the molding machine.
6. Determine the molding conditions.
7. Set the molding parameters.
8. Run the analysis.
9. Review the results.
10. Solve filling problems.
11. Balance the runners.
12. Determine the packing profile.

At many of these steps, there could be iterations within the step, or iterations back to earlier steps.

Additional steps are required when considering cooling and warpage. Cooling is normally done before packing is optimized. While solving warpage problems, an iteration may require you to import a revised part model which, in effect, means running through the entire sequence again.

CHAPTER 8

Model Requirements

Aim

The aim of this chapter is to understand the requirements of a good midplane, Fusion and 3D model.

Why do it

To obtain accurate flow, cooling, and warpage results, it is critical to begin with a good finite element model. By knowing what makes a good model for finite element analysis, you will be more likely to create good models in your CAD system, and translating that model for use in MPI will be easier.

Overview

In this chapter you will look at various parameters used to measure mesh quality. These will include items from the mesh statistics including:

- Free edges.
- Non-manifold edges.
- Mesh match ratio.
- Aspect ratio.
- Connectivity regions.
- Mesh orientation.
- Intersections.
- Overlaps.

Specifically for a Fusion model, you will also look at how mesh density and defined thickness affect the flow analysis. For 3D models, there are specific mesh quality attributes that must be met also, including:

- Inverted tetras.
- Collapsed faces.
- Insufficient refinement through the thickness.
- Internal long edges.
- Tetras with extremely large volumes.
- Tetras with high aspect ratios.
- Tetras with a small angle between faces.

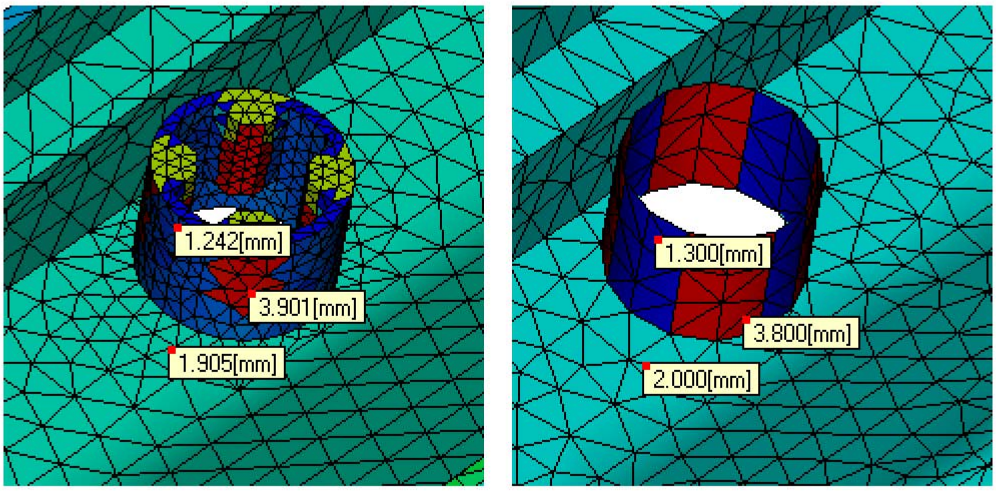
Theory and Concepts - Model Requirements

The requirements for Midplane, Fusion and 3D models are, in many cases, the same. Often though, a Fusion model has more model requirements than a Midplane model. 3D models are often built from a Fusion model, however, not all the requirements on a Fusion model apply to 3D models.

A Fusion mesh is a collection of 3-noded triangular elements describing the surface of the part. This is referred to as a Dual Domain mesh. It is sometimes called a double-skinned mesh. The distance between the elements determines the thickness of the part. The thickness is calculated automatically in Synergy and the analysis solvers. However, Synergy does allow you to specify a thickness value and thereby override the automatically calculated value.

A Midplane mesh consists of a web of 3-noded triangular elements that forms a 2D representation of a solid model. The elements are normally on the center line, or mid-plane of the part's cross-section. Part thickness is specified as a property of the element, and needs to be set for each element. Depending on how the Midplane model has been constructed, you may need to manually set the thickness of the elements. Figure 85 shows the same part represented as both a Fusion and Midplane model.

The Midplane and Fusion flow solvers are very similar. The main difference is in how they calculate the thickness of the part.



Fusion Model **Midplane Model**

Figure 85: Fusion and Midplane representations of a part

On the surface, a 3D mesh looks like a Fusion mesh. A 3D mesh, however, consists of 4-noded tetrahedral elements arranged in layers through the thickness of the part. A tetrahedral mesh does not require a thickness property as it is a true volume-filling mesh. A 3D mesh can be created from a Fusion mesh in Synergy, or can be created directly when importing certain CAD file formats.

General mesh requirements

Below is a detailed description of the requirements. 3D meshes are often made from Fusion meshes, so the mesh requirements for a 3D mesh are the same as for a Fusion mesh, unless otherwise stated.

Edges

There are 3 types of edges a finite element model can have:

- Free edges.
- Manifold edges.
- Non-manifold edges.

Free edges

A free edge is an element edge that is not shared with any other element. A Fusion mesh must not have any free edges. Free edges are displayed in red on the Free Edges Diagnostic plot. Figure 86 shows a free edge on a Fusion model. In this case, the free edge is caused by a rib that is not connected to the base of the part.

A Midplane mesh will have many free edges, for example, along the parting line or around the edge of a hole. There may be cases where there is a free edge where there should not be one - you have to manually check for this.

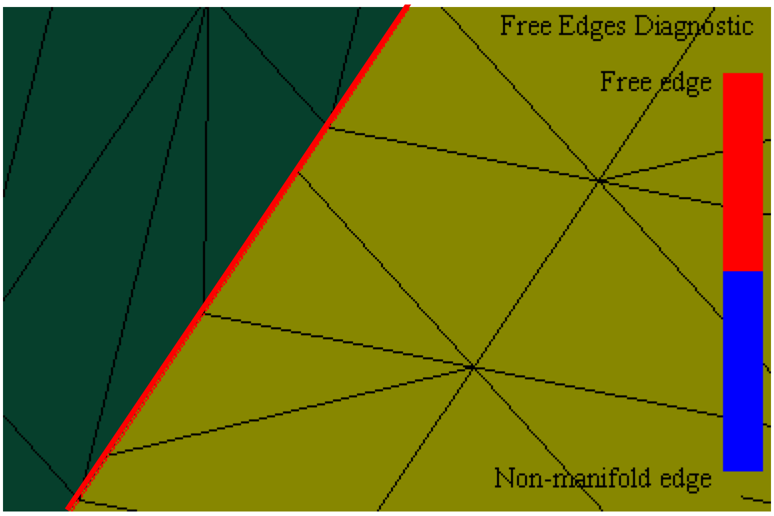


Figure 86: Free edge

Manifold edges

A manifold edge is an element edge that touches exactly one other element. This is the only kind of edge a Fusion model should have. Midplane models will also have many manifold edges. Both Fusion and Midplane models can have any number of manifold edges. The mesh statistics will list the number of manifold edges. The total number of manifold edges will depend on the size and complexity of the model. A 3D model's elements touch on the faces of elements not the edges so this does not apply.

Non-manifold edges

A non-manifold edge is an element edge that touches two or more other elements, for example, a "T" shaped cross section. Fusion models should not have any non-manifold edges, whereas Midplane models will have a non-manifold edge at every rib intersection and many other junctions. Non-manifold edges are displayed in blue on the Free Edges Diagnostic plot. Figure 87 shows a non-manifold edge on a Fusion model.

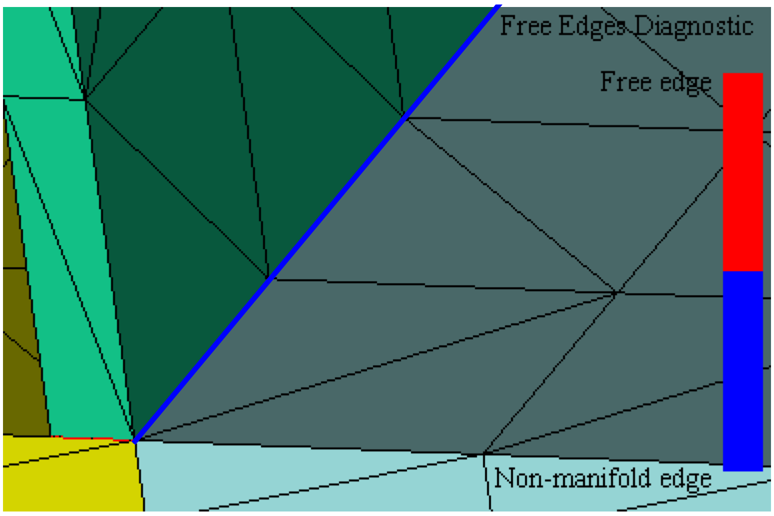


Figure 87: Non-manifold edge

Mesh match ratio

The mesh match ratio is a characteristic of Fusion models only. Figure 88 shows an example of a matched and unmatched group of elements for the same geometry. The matched elements will have a better thickness determination and lead to a more accurate analysis. In Figure 88 you can see a comparison between a mismatched mesh (left), and a matched mesh (right). Elements in a Fusion model should be matched to an element on the other side of the wall thickness. To obtain acceptable simulation results, the mesh match ratio should be above 85% for a flow analysis, and above 90% for a warpage analysis. If the mesh match ratio is not high enough, it is generally an indication that the mesh density is not high enough, or the part is a bit too chunky for Fusion. A chunky part is best represented by a 3D model.

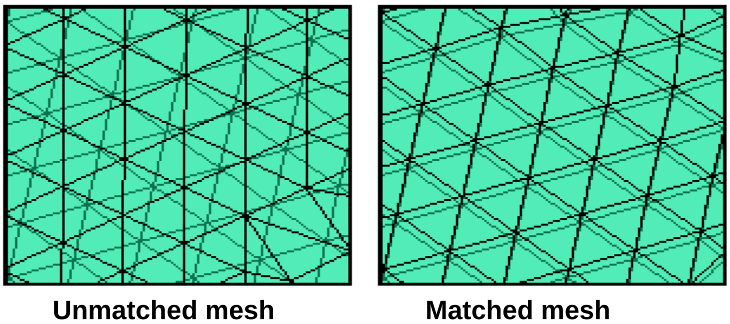


Figure 88: Mesh Match shown by using transparent elements

The Mesh Match Diagnostic plot in Figure 89 shows elements that are matched (blue), unmatched (red) and edges (green). Edge elements should only be elements that form the edge of the part. Unmatched elements are normally found on both sides of the outside corner of a part. If edge or unmatched elements are found elsewhere in the model, this may lead to problems in the simulation.

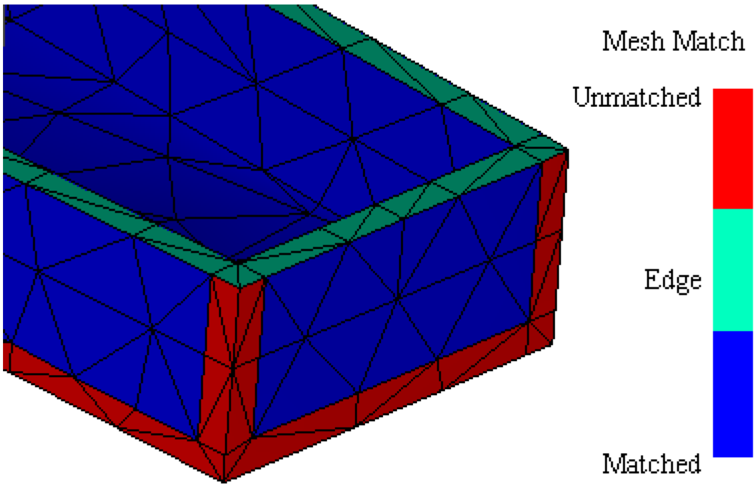


Figure 89: Match ratio diagnostic

Reciprocal match ratio

For warpage analysis, the **reciprocal mesh match** ratio should also be high, that is, above 90%. The reciprocal mesh match ratio is the percentage of all element matches where one element is matched to a second element and the second element is matched back to the first element, as shown in Figure 90. This may prove difficult to reach in models that contain ribs and/or curved surfaces. In some cases you may only reach 85%, but the higher the percentage, the better the result.

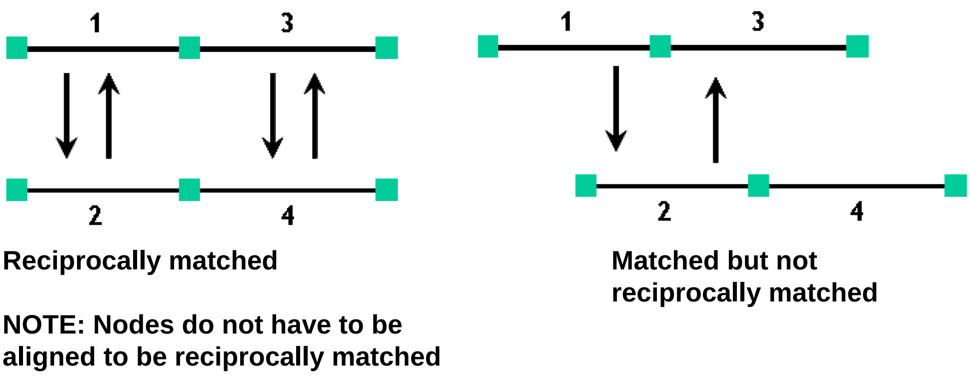


Figure 90: Reciprocal match

Aspect ratio

The aspect ratio for a triangular element is the ratio of its longest side (L) to its height (h), as shown in Figure 91. The lower the aspect ratio, the better. The average aspect ratio should be below **3:1**, and the maximum should be below **6:1** for Midplane and Fusion models. This is difficult to achieve in practice for complex Fusion models. When creating a 3D mesh from a Fusion mesh, the maximum aspect ratio before conversion should ideally be below **30:1**.

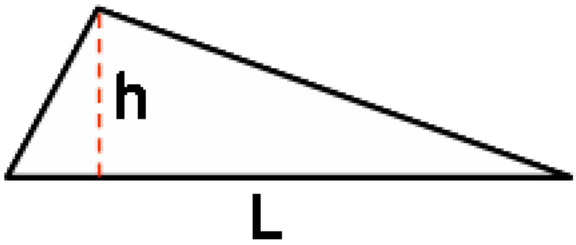


Figure 91: Aspect ratio

The higher the aspect ratios, the more probable they will have a negative impact on the analysis results. Flow analysis is the least sensitive to aspect ratios, whereas cooling and warpage analysis are far more sensitive. If the aspect ratios are too high, solver convergence problems can occur. This can lead to illogical results or, in severe cases, cause the analysis to fail.

Lowering the element aspect ratios is a very important objective in improving mesh quality. It is normally the most time consuming part of model preparation. Good CAD design will prevent the worst aspect ratio problems.

In Figure 92 below, the corner of the part has a small radius that is causing many high aspect ratio elements. This is a problem that should be fixed. In the Aspect Ratio Diagnostic plot, high aspect ratio elements are identified by a colored line projecting outwards perpendicular (normal) to the element face. The color of the line indicates the magnitude of the aspect ratio value, as displayed on the plot legend.

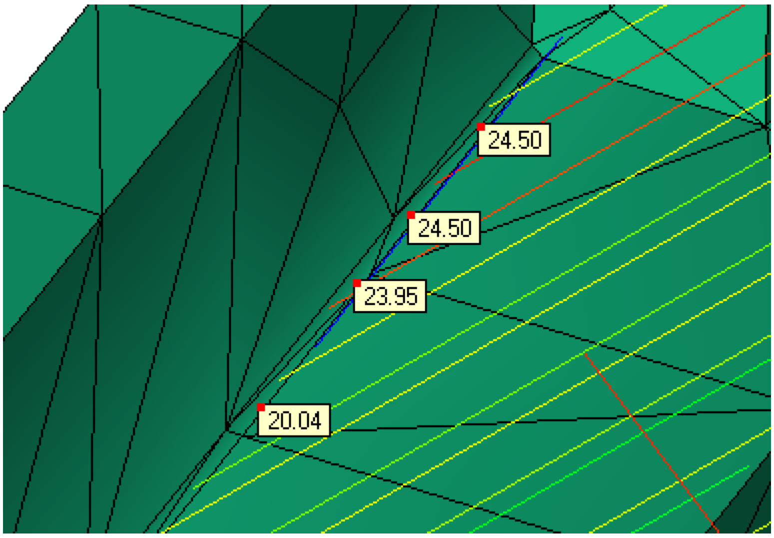


Figure 92: High aspect ratio elements

Connectivity regions

A connectivity region is a group of elements that are connected together. A Fusion, Midplane or 3D model must only have one connectivity region for the part as shown in Figure 93. For some CAD models, groups of entities may be disconnected from other parts of the model during model translation. This is a problem that must be fixed. The Connectivity Diagnostic plot shows the number of connected regions. A part model must only have one connectivity region.

When water lines or other cooling features are added, more than one connectivity region will be present. In this case you must account for each individual circuit in the count. Normally, the part model is cleaned up before water lines are added, so you will know that there is only one connectivity region.

```
Entity counts------------------------------
    Surface triangles              2812
    Nodes                          1402
    Beams                          0
    Connectivity regions           1
    Mesh volume                    8.874 cm^3
    Mesh area                      123.974 cm^2
```

Figure 93: Connectivity regions count in the Mesh Statistics

Mesh orientation

Orientation determines the top and bottom side of an element. Some CAD systems call it defining the normal. The orientation for midplane models needs to be consistent for viewing results. Element orientation can be checked using the **Orientation Diagnostic** plot. In the Orientation Diagnostic plot, each element is colored according to its orientation: blue for the top side; and red for the bottom side. Figure 94 shows the orientation of a Fusion model. For Fusion models, the outside of the model must always be the top side (blue). In this case, there is one element that is not oriented correctly. The Orient Elements command is a Global Mesh Tool found in the toolbox, can be used to fix orientation problems.

Tetrahedral elements are space-filling and therefore do not require orientation. When generating a 3D mesh from a Fusion mesh, you should always first ensure that the Fusion mesh is oriented correctly.

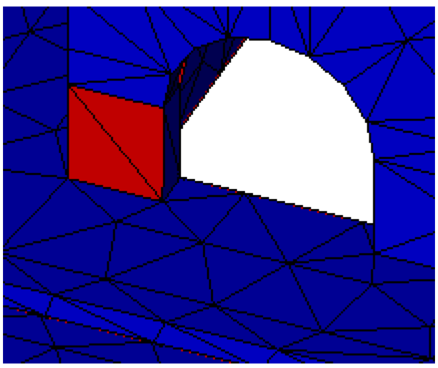


Figure 94: Mesh orientation for a Fusion model

Intersections

The mesh diagnostics distinguish between two types of intersection-related errors: intersections and overlaps. An intersection is when one element passes through the plane of another element as shown in Figure 95. An overlap is when two elements are in the same plane, and their faces fully or partially overlap as shown in Figure 96.

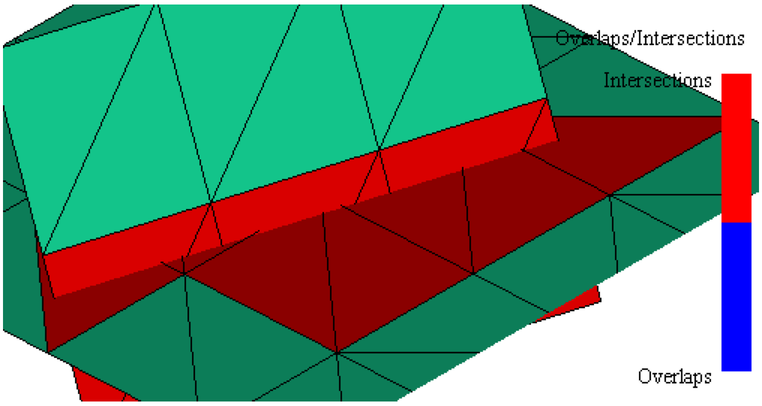


Figure 95: Intersections

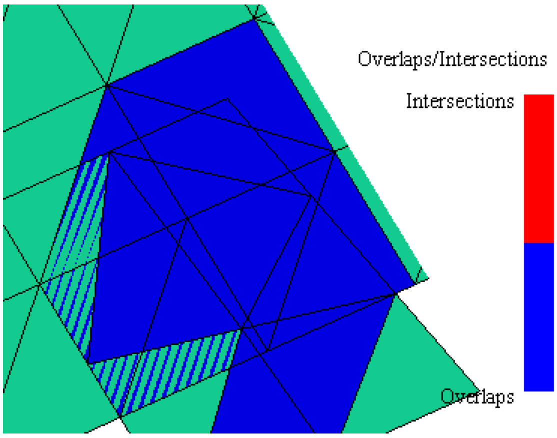


Figure 96: Overlaps

When importing CAD geometry, it is important to ensure the geometry is good and not corrupt, as corrupted CAD geometry often leads to a poor mesh. Should this be the case, most intersections and overlaps can be fixed with the Mesh repair wizard.

Zero area elements

Sometimes when importing a poor quality CAD model, or in the process of cleaning up a mesh, zero area elements are created that look like a line, as shown in Figure 97. The Zero Area Elements Diagnostic plot highlights the nodes on the element. The zero area element diagnosis is based on a user-specified edge length tolerance. It is possible for a very small equilateral triangle to be identified as a zero area element as shown in Figure 98. These elements can be caused by very small detail on the part or mesh cleanup problems. Small elements should be deleted.

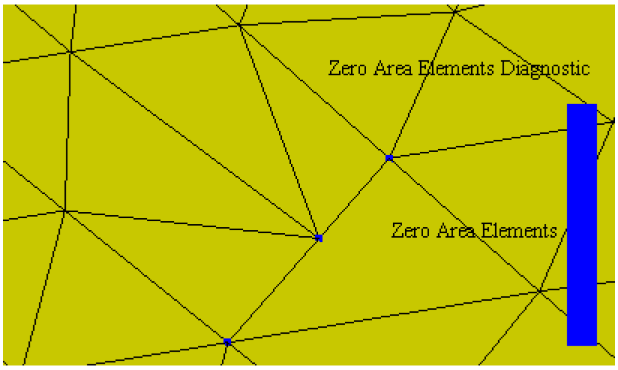


Figure 97: Zero area element caused by a collapsed element

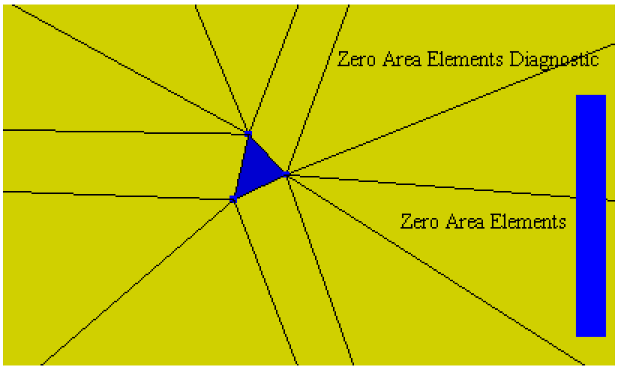


Figure 98: Zero area element caused by a very small element

Thickness representation

For both Fusion and midplane meshes, it is very important to ensure that the thicknesses in the part are accurately represented. For a midplane mesh, thickness values have to be assigned explicitly for midplane meshes. For Fusion meshes, however, thicknesses are calculated automatically. There is an option to manually assign thickness and override the automatically calculated value. Figure 99 shows the cross-section of a part with a dotted line representing the center line of the main component. Figure 100 shows the automatically calculated thickness for a Fusion mesh. The cross-section of this part is very "chunky", therefore the thickness in the part is not represented well. This geometry is best represented by a 3D tetrahedral mesh. The mesh matching on this part is not good. The mesh match ratio may have a high percentage, but the matching does NOT represent the thickness well at all. The 6 ribs on the sides and top of the part are wider than they are high. In all of the ribs, the thickness of the rib has been considerably over-estimated.

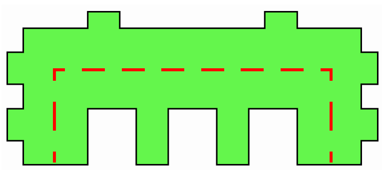


Figure 99: Part cross-section

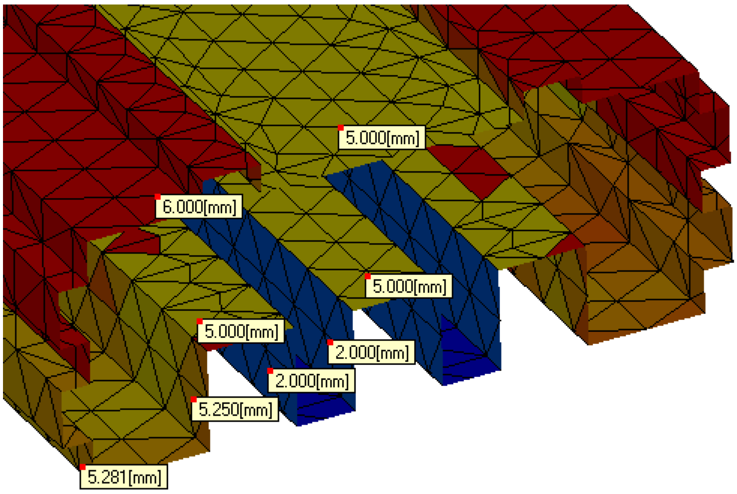


Figure 100: Thickness representation of a part

For Fusion parts with more "shell" like geometry, the thickness should still be checked and modified if necessary. When assigning thickness values to edge elements, a thickness of 75% of the face should be used. For example, if the nominal wall of a part is 2.0 mm, the edge elements connecting the top and bottom faces should be assigned a thickness of 1.5 mm.

Chunky geometry

A part or portion of the part is considered to be chunky based on its width to thickness ratio. Hele-Shaw model used for midplane and Fusion models assume there is no heat transfer from the edge of the element. For a ratio of 10:1 only 9% of the perimeter is the edge. When the ratio drops to 4:1 the perimeter percentage is 20. When the ratio is 2:1, 33% of the perimeter is the edge. Experimentation has determined that there is a significant loss in accuracy if the ratio drops below 4:1. Triangular elements should not be used in these situations. Beam elements can be used, or a 3D tetrahedral element can be used.

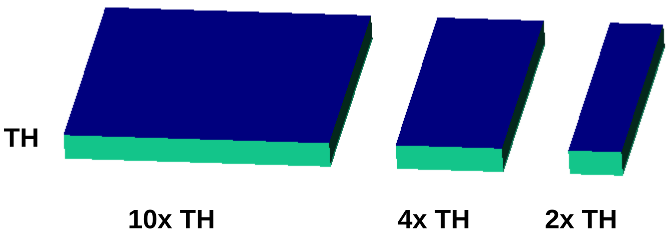


Figure 101: Thickness to width ratios

3D Mesh tetrahedral mesh specific requirements

Inverted tetras

A tetra (tetrahedral) element has four triangular faces. Each face can be shared by another tetra. The nodes of the two tetras not on the face must not be on the same side of the face, otherwise they are called inverted tetras. Figure 102 shows an example of two tetras that are correct and then not correct.

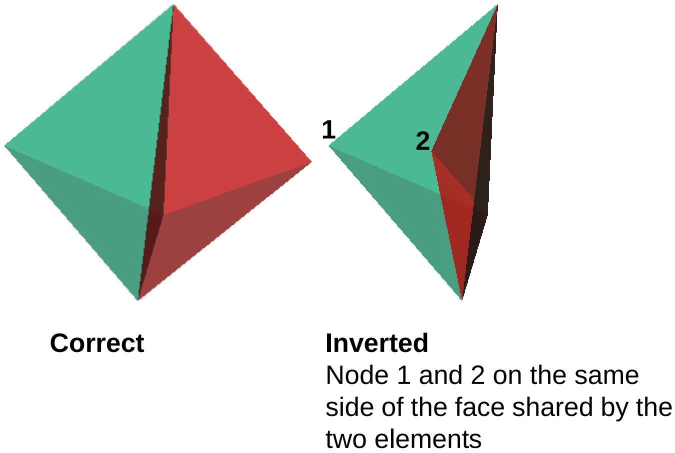


Figure 102: Inverted elements

Collapsed faces

A collapsed face occurs when a node is used two faces or surfaces of the part. The local thickness is zero. This cannot occur and must be fixed. Figure 103 shows an example.

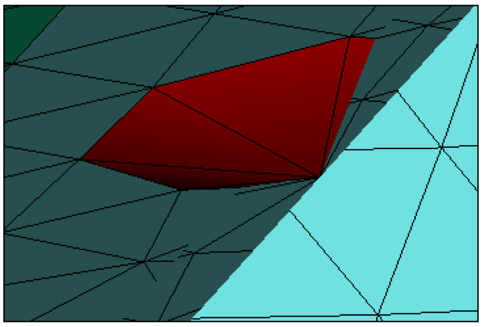


Figure 103: Collapsed faces

Number of tetrahedral element layers

For a 3D tetrahedral mesh, there should be at least 6 layers of elements through the thickness. An example is shown in Figure 104. If the material is fiber filled and a fiber flow analysis is going to be used, then a minimum of 8 layers should be used. To keep the aspect ratio low enough, the global edge length should be two times the wall thickness at a maximum.

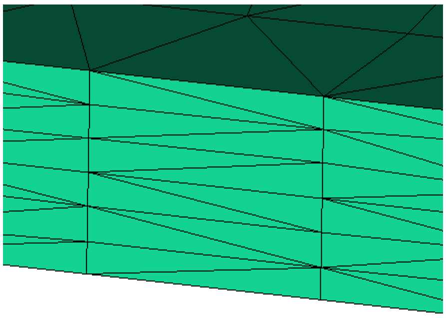


Figure 104: Layers through the thickness of a 3D tetrahedral element model

Internal long edges

An internal long edge is an edge that is too long compared to the average edge length on corresponding surface. Edge length ratio, maximum internal edge length at an internal node vs. average surface edge length in the local area, is used as the criteria and its default value is 2.5. Ratios greater than 2.5 should be eliminated

Extremely large volumes

Elements that have a volume of more than 20 times that of the average element volume are considered to have a large volume. These elements must be broken up and made smaller.

High aspect ratios

The maximum aspect ratio for a tetrahedral element is 50:1. Preferably, the aspect ratio should be lower. When converting from a Fusion mesh to a 3D, a lower the surface mesh's aspect ratio will tend to help lower the tet mesh aspect ratio.

Small angle between faces

Nodes on tetrahedral elements should have similar spacing between the 4 nodes. When one node of an element is close to the plane formed by the other three elements the angle between the faces gets very small, as shown in Figure 105. Elements with small angles cause convergence problem. The minimum angle is 2 degrees. Small angles become a bigger issue the more layers that are defined. The global edge length needs to get smaller as the number of layers increases to prevent this issue.

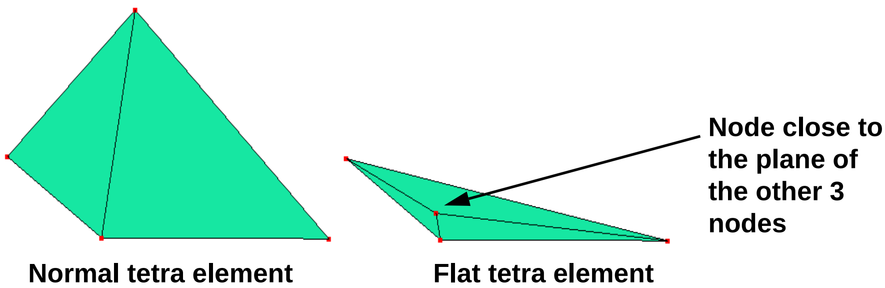


Figure 105: Tetra element with a small angle between faces

Mesh requirements summary

The mesh requirements described above for Fusion, 3D and Midplane mesh types are summarized in the following table.

Table 16: Summary of mesh requirements for the three mesh types

Mesh Issue	Fusion	3D	Midplane
Free edges	Must NOT have any	Same as Fusion	Are valid at boundary of holes and parting line.
Non-manifold edge	Must NOT have any	Same as Fusion	Are valid at "T" cross-sections. e.g. ribs
Manifold edge	Only type of edge allowed	Same as Fusion	Same as Fusion
Match ratio	> 85% for flow, > 90% for warpage	Not Applicable	Not Applicable
Reciprocal Match	> 90% for warpage	Not Applicable	Not Applicable
Aspect ratio	Average < 3:1, Maximum <6:1	< 30:1 on Fusion mesh before conversion	Same as Fusion
Connectivity regions	One group for the part	Same as Fusion	Same as Fusion
Element orientation	The top (blue) side of the element pointing outward	Same as Fusion before conversion	Consistent mesh orientation
Intersections	Must NOT have any	Same as Fusion	Same as Fusion
Overlapping elements	Must NOT have any	Same as Fusion	Same as Fusion
Zero area elements	Must NOT have any	Same as Fusion	Same as Fusion
Thickness representation	Must have thicknesses properly modeled	Not Applicable	Same as Fusion
Inverted tetras	Not Applicable	Must NOT have any	Not Applicable
Collapsed faces	Not Applicable	Must NOT have any	Not Applicable
Number of 3D layers	Not Applicable	6 layers typically OK, 8 better for fiber orientation	Not Applicable
Internal long edges	Not Applicable	< 2.5:1	Not Applicable
Extremely large volumes	Not Applicable	<20:1	Not Applicable
High aspect ratios	Not Applicable	< 50:1	Not Applicable
Small angle between faces	Not Applicable	> 2 degrees	Not Applicable

Mesh density considerations

It is generally easy to achieve a mesh density that can provide good pressure predictions. It does not take a fine mesh to accurately predict pressures. Filling effects, however, can only be accurately predicted if the mesh is detailed enough to capture relevant details of the model. Three important considerations include:

- Hesitation.
- Air traps.
- Weld lines.

These issues represent common mesh density related problems. If the mesh is not fine enough, the analysis will not pick up these problems.

Hesitation prediction

Hesitation is a slowing down of one area of the flow front compared to another. To some degree, a small amount of hesitation can be designed into the mold, as is done when flow leaders or artificially balanced runners are used. However, to pick up these or any other type of hesitation effects, a fine mesh is required. The top portion of Figure 106 shows the effect on the predicted filling pattern when the mesh is not fine enough. The center section of the part is 1mm thick, the top is 2 mm and the bottom is 3 mm. Clearly, with the coarse mesh there is no lagging in the thin middle section. The bottom portion of Figure 106 there is at least 3 rows of elements across each change in thickness. A much better hesitation pattern is evident in the predicted flow front.

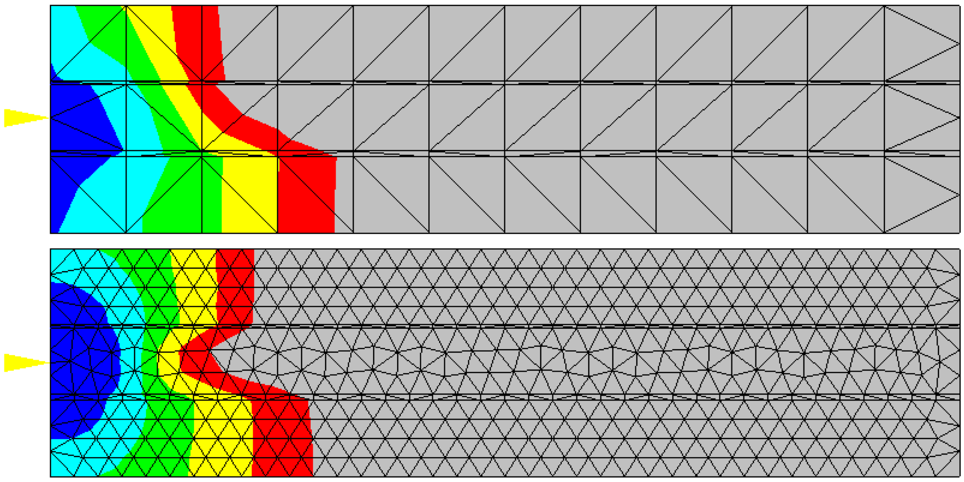


Figure 106: Hesitation prediction with a fine and coarse mesh

> To ensure that hesitation effects are correctly predicted, there should be at least 3 rows of elements across any major change in thickness.

Air trap prediction

Air traps on a part are often caused by hesitation due to changes in wall thickness. The prediction of air traps will only be as good as the mesh density allows. With a coarse mesh in a thin area, air traps will not be predicted or displayed. With a fine mesh air traps are predicted. In Figure 107 below, the nominal wall is 2.5 mm and the thin wall is 1.25 mm. The thin area in the center of the part. Notice with the coarse mesh that no hesitation is predicted in the thin section, whereas hesitation is predicted with the fine mesh. This is shown by the relatively straight contour lines through the thin area in the top portion of Figure 107, while in bottom portion, the contour lines encircle a node.

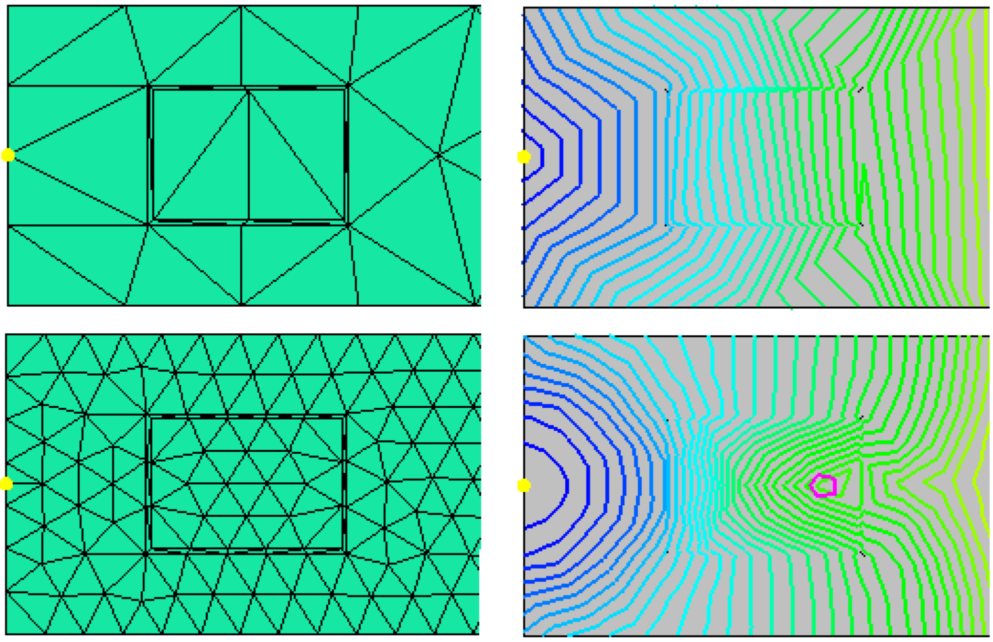


Figure 107: Air trap prediction with a coarse and fine mesh

To ensure that gas traps are correctly predicted, there should be at least 3 rows of elements in thin areas of the part.

Weld line prediction

A weld line is formed when the polymer flow front separates and comes back together. This occurs around holes, and with multiple gates. In a simulation, weld lines are formed at nodes. When a weld line is predicted at two or more connected nodes, a line is drawn between the nodes. Weld line prediction is very sensitive to mesh density issues. Therefore, when weld line information is required, a fine mesh is essential as a coarse mesh does not always indicate the presence of weld lines. In the top portion of Figure 108, the mesh is coarse. The resulting filling pattern predicts a weld line around the first hole, but not the second due to the mesh density. In the bottom portion, the weld lines are predicted around both holes.

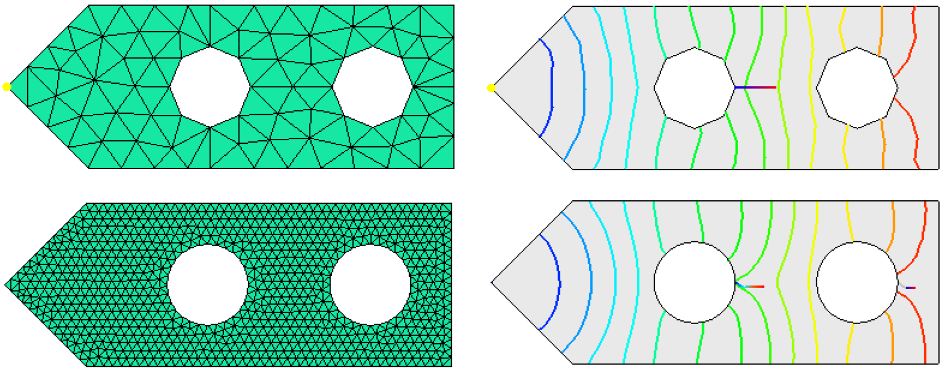


Figure 108: Weld line prediction with a coarse and fine mesh

When reliable prediction of weld lines is critical, ensure those areas of the part where they are most likely to occur are finely meshed.

Part details

To properly represent a plastic part for flow analysis, there are 3 characteristics of the part that need to be modeled accurately:

- Thickness.
- Flow length.
- Volume.

When these characteristics of the part are modeled accurately, the flow analysis will be accurate.

Thickness

The wall thickness of the plastic part is the largest contributor to the pressure drop in the part. It is the most critical characteristic of the part design to model for flow analysis. For a Fusion model, the thicknesses are calculated automatically by default. The distance between matched elements determines the thickness. For a Fusion model, elements on the edge of the part are set to 75% of the thickness of face elements touching the edge. If the thickness of a Fusion model does not accurately represent the thickness of the part, try re-meshing the part with a lower global edge length value. The thickness of elements can also be set manually. This should NOT be done if the model will then be used for cooling and warpage as these solvers do not support manually assigned thicknesses.

The wall thickness of the plastic part is the largest contributor to the pressure drop in the part.

Flow length

The flow length in the part is the second most important characteristic to model for flow analysis. The combination of wall thickness and flow length will determine the pressure required to fill a part. For midplane, Fusion and 3D models, the flow length is not explicitly calculated for a fill or flow analysis, but is for the molding window analysis. In the case of a fill or flow analysis, as the flow front expands and moves further from the gate, the flow length is calculated dynamically and recorded and used internally.

Volume

The volume of the part is calculated from the part shape, size, and wall thickness. The accuracy of the volume calculation can be used gauge whether a midplane model has been modeled accurately. Normally, the target is for the calculated volume to be within 5% of the true volume. The calculated volume for Fusion models will generally be more accurate due to the surface mesh. The volume is important as it helps define the flow rate needed in the part, and will significantly influence the pressure calculations in the runner system. The volume of the part has little influence on the pressure drop within the part itself. When the runner system is added, the part volume will influence the flow rate in the runners and therefore the pressure drop.

Comparing thickness and flow length

The graph in Figure 109 below summarizes the results from a series of analyses where the thickness, flow length and were changed to see the effect on pressure. The parameters where changed in increments of 20%. The thickness was reduced from 4.5 mm to 1.8 mm. The flow length was increased from 100 mm to 180 mm. The thickness decreased from 125 mm to 25 mm. The parameters were changed so the pressure would increase from the base model. In each case where the thickness or flow length changed, the volume of the part stayed the same, 11.25 cm^3 by adjusting the width. The material was a nylon, and the processing conditions did not change for any of the analyses.

It is clear from the graph that thickness has by far the greatest influence on the percent change in pressure. Changing the volume has virtually no effect on pressure. The differences in percent change between flow length and thickness may change a little with different processing conditions and materials, but thickness will always have the most effect.

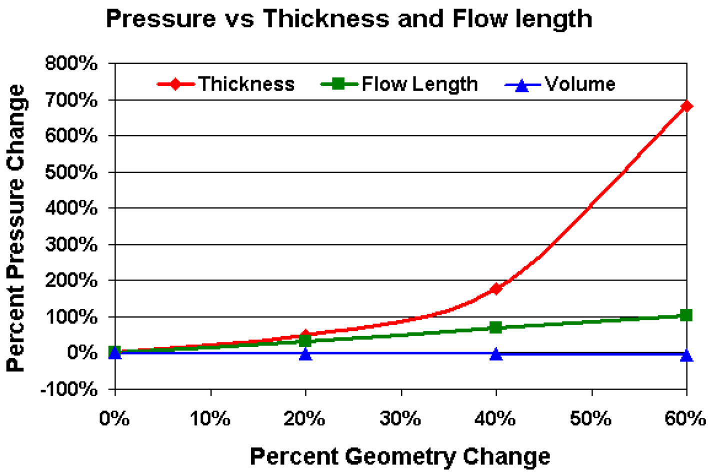


Figure 109: Effect of thickness, and flow length on pressure

Small radii

For both Midplane and Fusion models, the process variables (pressure, temperature, shear stress, shear rate, etc.) are calculated from one node to another as the flow front is propagated at a certain flow rate. As shown above in Figure 109, the biggest influences on the pressure drop are wall thickness then flow length. Small features such as corner radii that are significantly smaller than the nominal wall will have no impact on the analysis. Figure 110 shows a radius in the corner of a part. These are the types or features that are not needed for a model that will be using midplane and Fusion meshes. For 3D, they can be modeled, but generally don't have a big impact on the part.

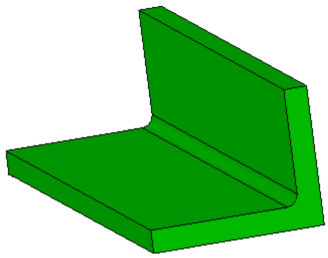


Figure 110: Small corner radius

What effect does a small radius have?

A small radius slightly increases the thickness. Consider Figure 111 and Figure 112 below. Figure 111 shows a part with a nominal wall thickness of 1.5 mm (0.059 in), a rib thickness of 1.0 mm (0.039 in), and a 0.25mm (0.010 in) radius in the corner. The dots represent nodes spaced a nominal distance apart for a Fusion model.

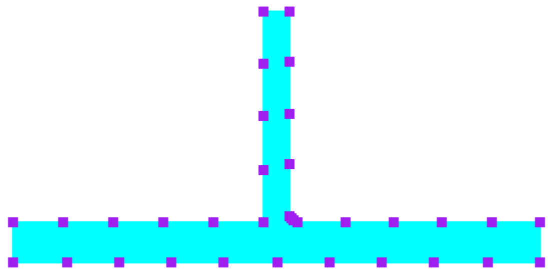


Figure 111: T cross-section

Figure 112 shows a magnified view of the rib/wall intersection, and the radius is now apparent. You can see that the radius is much smaller than the node spacing.

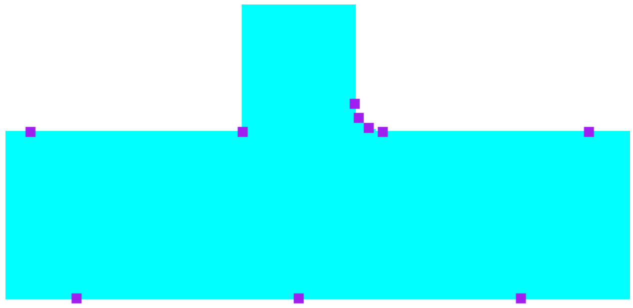


Figure 112: T cross-section corner

The **minimum** number of nodes that should be used if the radius is added is 3, but 4 would be better. With only 3 nodes, there are only two rows of elements across the radius. For a Midplane model, the thickness of the radius would change about 0.125 mm (0.005 in) for each of the two rows, and the width of each row of elements would be 0.125 mm (0.005 in). The resultant change in thickness or volume is insignificant. If this were a Fusion model, these small elements would not be matched, or would be matched incorrectly with another set of elements. The thickness would not be calculated correctly.

A bigger problem would be the very high aspect ratios caused by meshing the radii. With the nominal spacing of the nodes as shown, the aspect ratio in the corner is about 25:1, when it is preferable to keep the aspect ratio below 6:1. To fix this aspect ratio problem, there would need to be a much finer mesh density around the corner, thereby significantly increasing the number of elements. Assume you did not fix the aspect ratio problem. The flow solver takes much longer to converge on high aspect ratio elements than low ones. Adding the small corner radii adds little or nothing to the quality of the analysis, and significantly increases the compute time and/or causes convergence problems.

It is not possible to simulate shear around sharp corners in a Midplane, or Fusion model, due to the assumptions of the model. Because the flow analysis calculates from node to node, flow is within the plane of the element, or along the axis of a beam element. There is no provision in the calculations for going around corners, sharp or otherwise.

With a 3D flow analysis, however, the radii would affect the flow going around a corner. The shear rate is higher on the sharp corner. This is a "3D" effect that Fusion and Midplane do not pick up. Figure 113:shows the shear rate differences in a corner of a rib with and without a corner radius. The shear rate is a little higher in the corner without the radius, but it is only a very local influence on the part. To see the detail of the shear rate in the corner, a very fine mesh was required. This high corner shear rate has very little influence on the analysis as a whole with regards to pressure, temperature etc.

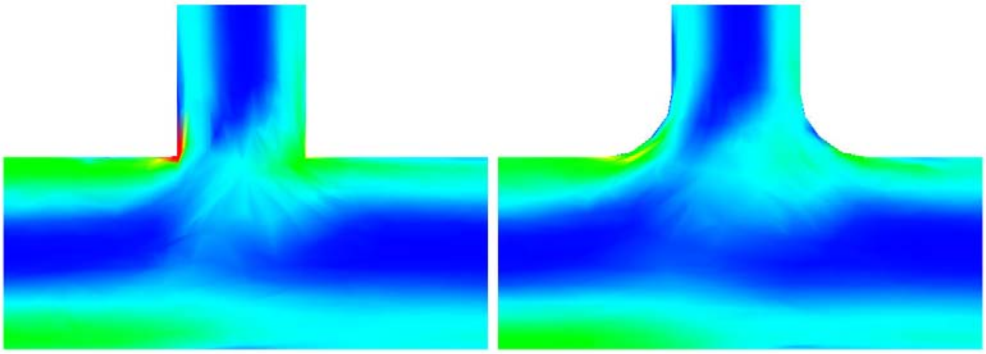


Figure 113: 3D shear rate in corner with and without radii

> The important thing to remember is you must think of the "**BIG**" picture. When modeling, the most important thing to get right is **thickness**, **flow length**, followed by **volume**.

Compute time - mesh density - accuracy

Compute time goes up exponentially as the number of elements increases. As can be seen in Figure 115, the pressure prediction for all of the models is within 7%, and within 3% for all models except the 2 smallest models. The compute time, however, goes up over 800 times between the smallest model, and largest model. If you exclude the first 2 models, the compute time still goes up 82 times between the model with 4,030 elements and the model with 58,750 elements. From the graph you can see that having a finer mesh than is necessary is a significant waste in time. There must be a balance between the mesh density and compute time. The mesh density must be sufficient enough to account for the changes in the geometry, but adding more elements than necessary just adds to the time required to get results without any gain in accuracy.

The part used to calculate the compute times and pressures is shown in Figure 114. The computer used was a 2.8 GHz PC with 1 GB of RAM. The software used was MPI 6.0.

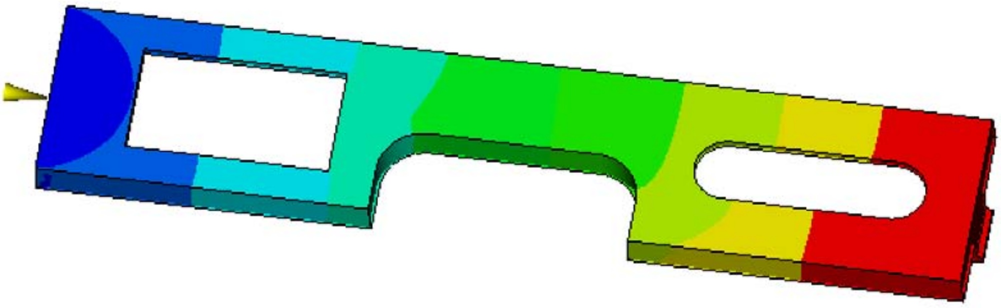


Figure 114: Part used for calculating compute time

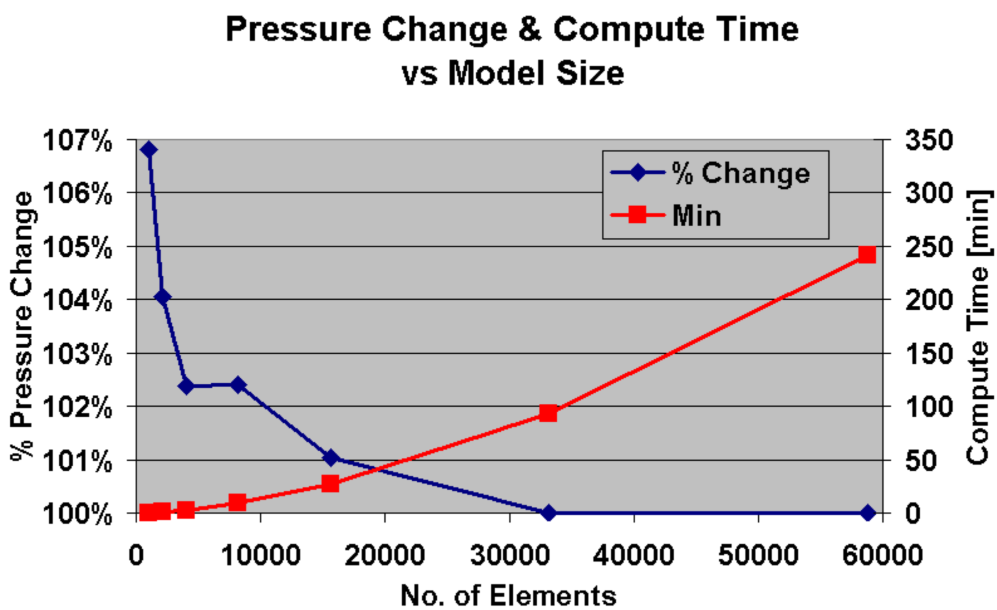


Figure 115: Pressure change and compute time vs. model size

What You've Learned

To obtain good flow analysis results, the mesh must be good. All mesh types must be free of mesh errors. In addition, meshes should have:

- Aspect ratios below the following limits.
  - 6:1 for Midplane and Fusion.
  - 30:1 for a Fusion mesh that will be converted to a 3D mesh.
  - 50:1 for a tetrahedral mesh.
- The Fusion mesh match ratio must be above 85% for flow analysis, and above 90% for warpage analysis.
- The thickness representation for Midplane and Fusion models must be accurate.

Thickness is by far the most important characteristic of a part to model to ensure accurate pressure predictions.

To pick up significant changes in thickness, there should be at least 3 rows of elements in such areas.

Once the above conditions are met, additional elements do not improve the accuracy, but the compute time will go up exponentially.

Small features like corner blends and radii add nothing to a Midplane and Fusion analysis except problems. For 3D they can be analyzed, but with a very high mesh density and little overall change in the results.

CHAPTER 9

Model Translation and Cleanup

Aim

The aim of this chapter is to learn how to import a model from a CAD system, check the model for errors, and then clean up any mesh problems that may be present.

Why do it

There are many ways to get geometry into MPI for **running** an analysis. Most of them involve taking a model from a CAD system, reading it into MPI, meshing it if necessary, checking the mesh quality, and finally fixing any mesh problems. The majority of the models analyzed in MPI are imported using this method.

Overview

Translation and cleanup of a model involves these basic steps:

1. Import a CAD model in one of many usable formats.
2. Mesh the imported geometry.
3. Check the mesh for errors.
4. Use the Mesh Repair Wizard and mesh cleanup tools to fix the model.

These steps will be discussed in detail in this chapter. This chapter focuses on Fusion primarily but also discusses how the translation process is different between Fusion and midplane or 3D. The extra steps involved for preparing 3D or midplane models is also discussed.

Theory and Concepts - Model Translation and Cleanup

Preparing a finite element mesh

The specific steps for creating meshes for analysis in MPI are listed below. Figure 116 shows these steps in a flow chart. These steps are described in detail below.

- Prepare CAD Model.
- Import CAD Model.
- Set Mesh Densities.
- Generate mesh.
- Evaluate mesh.
- Cleanup mesh.

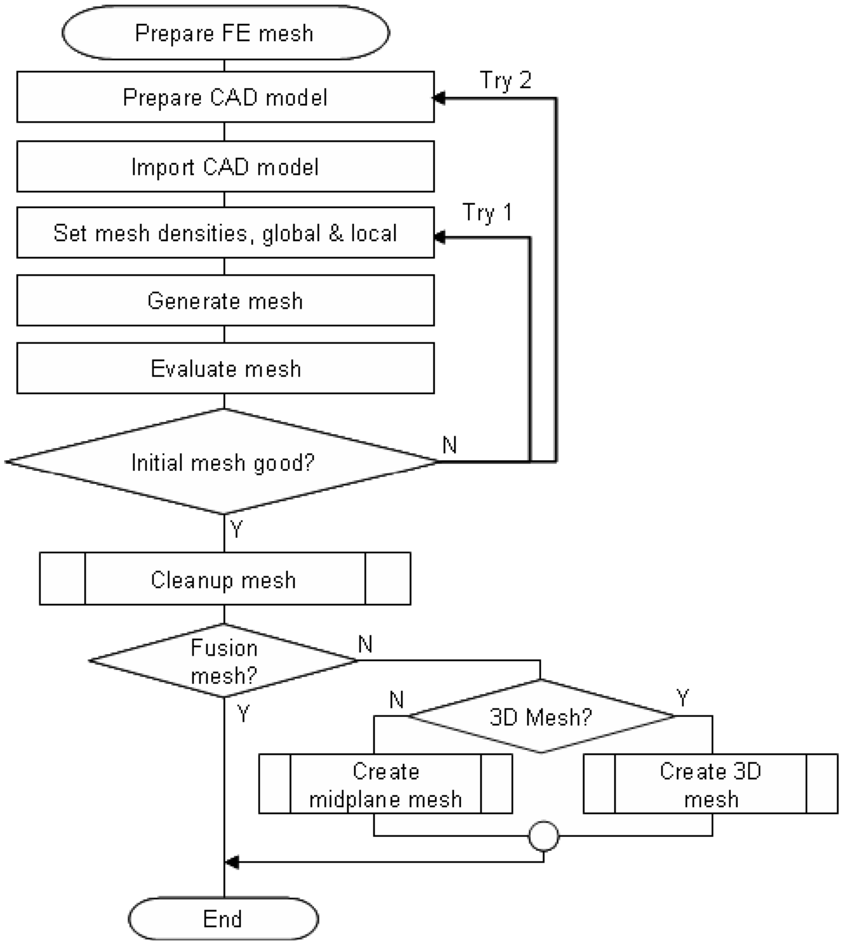


Figure 116: Preparing a finite element mesh flow chart

Prepare the CAD model

Most models analyzed in MPI use a CAD model as their source. This geometry must be free of errors. If errors exist in the CAD model, this will make the import more difficult. Moldflow has two products that can help with the validation and cleanup of CAD models. These programs are, Moldflow Magics STL Expert and **Moldflow CAD Doctor.** File formats supported by MPI come in a variety of formats described below.

File Format Categories

Many model formats can be read into MPI. These formats include six categories.

Category One

This category includes formats where model data comes in and a mesh needs to be created by Synergy. This is the most commonly used way of getting model data into MPI. Formats in this category include:

- Stereo-lithography (.stl).
- IGES (.igs, .iges).

Category Two

The second category contains formats that are already finite element meshes, and can possibly be used with little or no cleanup after importing. These formats include:

- IDEAS Universal, mesh only (.unv).
- ANSYS Prep 7 (.ans).
- Nastran (.nas).
- Nastran Bulk Data (.bdf).
- Patran (.pat).
- Fem (.fem).
- Moldflow (.mfl).
- C-Mold (.cmf).

Category Three

In this category, a special translator called "**Moldflow Design Link/Parasolid**" is required. This translator requires a special license. The translator provides added capability to read in the following geometry formats:

- Parasolid (.x_t, .x_b, .xmt_xmb, .xmb, .xmt).
- STEP AP203 (.stp, step).
- Iges (.igs, .iges).

Category Four

The fourth category requires a "**Moldflow Design Link/ProEngineer**" license. This license provides capability to read the following geometry formats:

- Pro/Engineer part files (.prt).
- STEP AP203 (.stp, step).
- Iges (.igs, .iges).

Category Five

The fifth category requires a "**Moldflow Design Link/CatiaV5**" license. This license provides capability to read the following geometry formats:

- CatiaV5 part files (.CATPart).
- STEP AP203 (.stp, step).
- Iges (.igs, .iges).

Category Six

The sixth category requires a "**Moldflow Design Link/Solidworks**" license. This license provides capability to read the following geometry formats:

- Solidworks 2005 part files (.sldprt).
- STEP AP203 (.stp, step).
- Iges (.igs, .iges).

File Format Considerations

STL Files

STL files are one of the most popular import file formats used to import models into MPI. An STL file is a form of triangular mesh. However, the mesh is not suitable for use in simulation. It needs to be re-meshed. It's the primary advantage of the STL format is that all solid modeling CAD systems export STL files and it has become an industry standard format to use. Also, Because an STL is a form of a mesh, it approximates curved regions with faceted surfaces.

Cad System Export Options For STL Files

Depending on the CAD system being used, there are many different options that can be used to generate STL files. In general, you should choose the settings that will give you the coarsest model without losing important detail. This is where the output can be controlled. Having a fine resolution of the curve can add significant unneeded detail to the part, as shown inFigure 118. Generally a low resolution is needed here. Depending on the CAD system, some of the controls for an STL resolution are:

- Angle control.
- Facet deviation.
- Chord Height.

The formula for determining chord height is:

$C = M/(1000 \times Q)$

C = Chord Height

p = Part surface

t = Tessellated surface

M = Diagonals of part bounding box

Q = Part quality (recommended 0.3, limits 0.1 to 1.0)

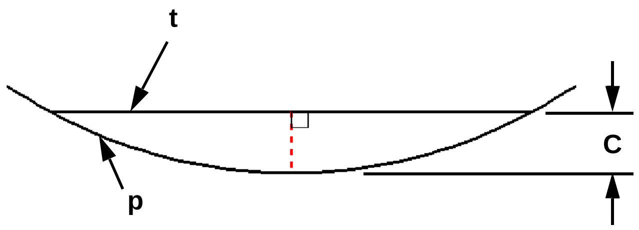


Figure 117: STL chord height definition

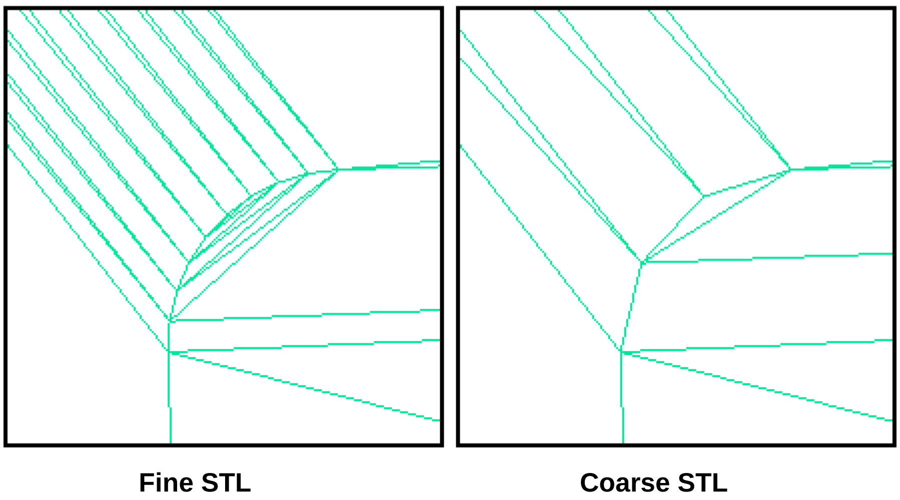


Figure 118: STL facet settings

All Geometry Formats

MPI can import complex geometry and retain the surfaces from many different formats. Although surfaces cannot be constructed in MPI, it can work with imported geometry in the following ways:

- View the imported geometry.
- Diagnose problems with the imported geometry.
- Set different mesh densities on individual entities.

Imported geometry-based models can be used to create Midplane, Fusion or 3D mesh types. If possible, simplify the model to remove unnecessary detail such as reference planes and very small features that have no effect on a flow analysis. This will result in a model that is more accurate with fewer elements.

IGES Model Translation

In order to successfully translate an IGES file in MPI, the following guidelines should be observed:

- The entire model must be described by IGES surfaces, not just lines and curves.
- Export from the CAD system as surfaces (not shells).

Refer to the on-line help topic **Supported entities, IGES, on the index tab** for more information on all supported IGES entities in MPI. IGES files can be imported directly into MPI, or MDL can be used. MDL imports more entities and has more flexibility during the import.

Meshed Formats

There are several formats that can be imported that already contain meshes. All of these formats support Midplane models – some support 3D and Fusion.

Legacy Formats

MPI will support formats from previous versions of Moldflow and C-Mold including:

- Moldflow MPI 2.0 (.mfl).
- C-Mold CMK2000 (.cmf).
- C-Mold (.fem).

IDEAS Universal File

The SDRC/I-DEAS universal file has the extension *.unv. The files can be used with Midplane, Fusion and 3D analysis.

Table 17 below lists the universal data sets that are recognized and translated when reading in a *.unv file. If the file contains data sets not listed here, they will be ignored in the translation process.

Table 17: Universal File Data sets

Data set	Description
151	Header
164	Units
2411	Nodes
2412	Elements
2429	Permanent Groups
2437	Physical Properties
2448	Physical Properties

ANSYS Prep 7 File

ANSYS model files have the extension *.ans. Table 18 shows a list of the supported entity types for ANSYS Prep 7 supported by Moldflow Plastics Insight. Table 20 lists the supported element types.

Table 18: ANSYS Prep 7 Supported Entities

Entity Type	Description
NBLOCK	Block formatted nodes
EBLOCK	Block formatted elements
EN,R5.0	Element card
EN,R5.1	Element card
EN,R5.5	Element card
EN,4.4	Element card
E,	Element card
EN,	Element card
R,R5	Real constant tables
R,	Real constant table

Table 19: ANSYS supported element types

Element Type
2 noded beam.
3 node triangle.
4 node tetra.
4 node quads converted to two 3 node tris.

Nastran Bulk Data FIle

NASTRAN Bulk Data files have the extension (*.bdf). Table 20 lists the NASTRAN element types that are supported by Moldflow Plastics Insight. All other element types will be ignored.

Table 20: Supported NASTRAN element types

Element type
GRID
CTRIA
CTRIA3
CQUAD
CQUAD4
CQUAD8
CQUADR
CQUADX

Table 20: Supported NASTRAN element types

Element type
CTETRA
CBAR (Beam elements)
PSHELL

PATRAN Neutral FIle

PATRAN Neutral files have the extension *.pat, or *.out. Table 21 lists the PATRAN element types supported by MPI.

Table 21: Supported PATRAN element types

Solid	Shell	2D
TET (4 noded)	TRI (3 noded)	BAR
	QUAD (4 noded)	

Defining Element Property Data

The element attributes required by Moldflow software can be passed in the second record of the Element Property Packet (04). These element properties can be set with the PFEG command in PATRAN. Table 6 summarizes the property data that the interface expects in each field.

Table 22: Property assignment

Field	Property Data
1	not used
2	thickness
3	not used

Import CAD model

Importing a model is just like a **File ➡ Open** command in most business applications such as a word processor. By default, the Import dialog will show all importable CAD file formats, as shown in Table 119. The **Files of type** box allows you to specify the type of file you are looking for.

The following steps are the typical steps needed to import geometry into MPI.

1. Click the icon  **File ➡ Import**.
2. Navigate to the directory where the CAD model is located.
3. Click on the file you want to import.
4. Click **Open**.

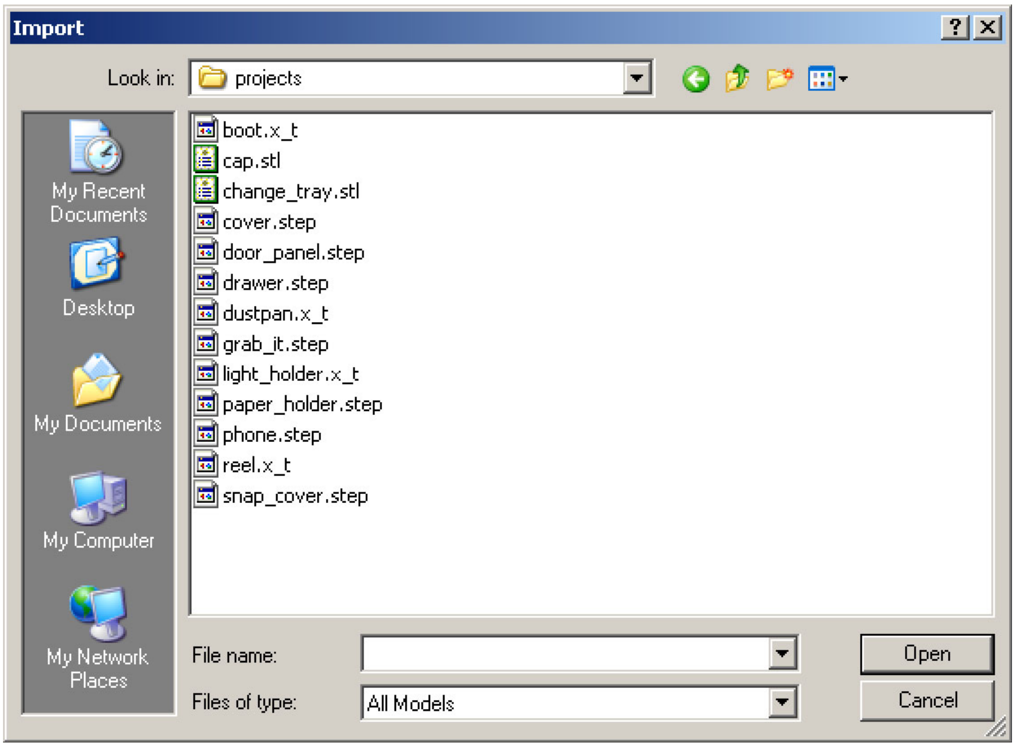


Figure 119: Import File Selection

In **Preferences** on the **General tab** there is a check box for the Import directory. When checked, the Import dialog will open in the project directory. When not checked, it will open in the last directory you imported a file from.

Study files can be imported from one directory to another. To import a study file, the **Files of Type** box must be set to (*.sdy).

Import dialog

The import dialog asks you to verify the type of mesh you wish to import. The choices include:

- Midplane.
- Fusion.
- 3D.

If you are importing a solid model always choose **Fusion**, as shown in Table 120. If you eventually want to work with a Midplane or 3D model, initially import the CAD model as Fusion. Study the model and clean it up as necessary. After the model is cleaned up, then the mesh type can be changed and re-meshed with the new desired format.

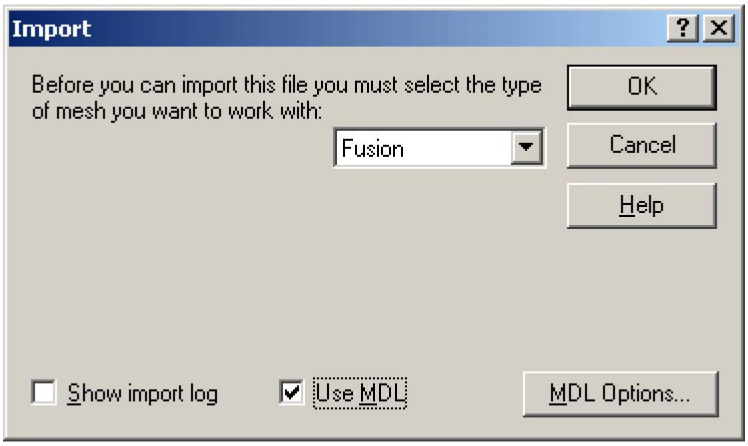


Figure 120: Set file type to Fusion for Geometry formats

Import geometry

Table 120 is an example of the import dialog for IGES. All other geometry formats including the following have the same dialog except for the Use MDL checkbox.

- STEP.
- Parasolid.
- Native Pro-Engineer.
- Native Catia.
- Native Solidworks.

These formats must use MDL to import the format. IGES can be used with or without IGES. When importing IGES, if the Use MDL box is not checked, the MDL Options button is not displayed.

Importing with MDL

For models that require MDL for importing, there is an **MDL Options** button in the lower right corner of the dialog, as shown in Table 120.

The MDL Import options, as shown in Table 121, control how the geometry is imported. In the **Settings** frame there are check boxes for **Generate mesh** and **Translate surfaces**.

Generate mesh checkbox

When the Generate mesh box is checked, the **Edge Length** frame appears. This gives you control over the mesh size when the part is imported. Enter a nominal edge length in the **Specified** field. When the mesh is created, it will use some adaptive meshing techniques that will help ensure a finer mesh in areas with small detail, and coarser meshes in wide-open areas. With MPI 6.0 this option is off by default. There is now much more control of the mesh density inside Synergy than there is with MDL. Normally, just the surfaces are translated. This is the default.

Kernel

The **Kernel** radio buttons are normally correctly clicked for the file type being read in. They relate to the file format being read in.

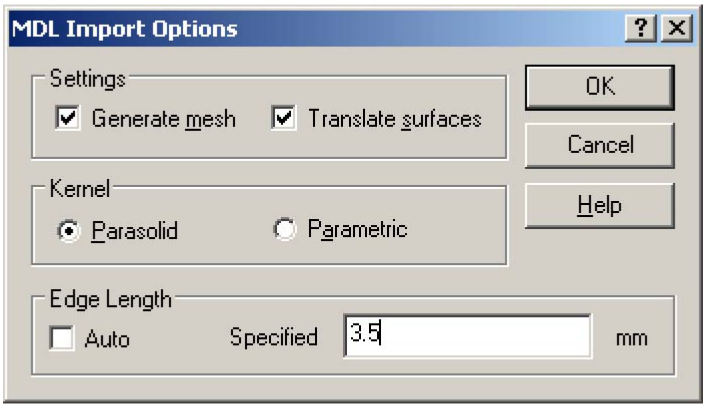


Figure 121: MDL Options

Generally, the best method of MDL import is to translate the surfaces and NOT to generate the mesh. There is more control for meshing in Synergy (MPI 6.0) than there is in earlier versions of MPI.

STL

When importing an STL model, there is an option for verifying the units of the model, as shown in Figure 122. With STL files the units of the file are not listed in the file. The approximate dimensions of the model are listed and the default units are based on those dimensions. If the default is not correct, change the units.

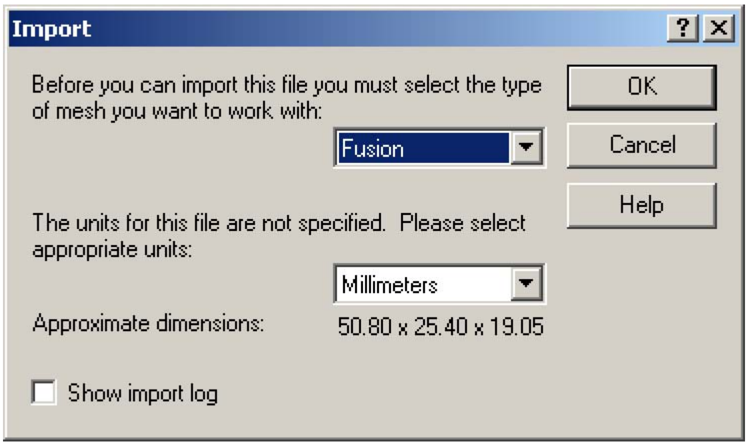


Figure 122: Import STL file dialog

Set Mesh Density

The mesh density can be set locally on individual surfaces imported or globally. The local mesh density is set before opening the generate mesh command.

Local Mesh Sizing

The mesh density of a part by default has a global setting. It is called the **Global edge length**. This means that the default mesh density would be the same for the whole part. MPI has the option of changing the mesh density of a part individual imported surfaces or regions created in Synergy. The mesh density of a part is changed with the command **Define Mesh Density** shown in Figure 123. In most cases, setting a local mesh density is done only after several global settings are tried and the mesh density is not sufficient in some local area. Then an existing mesh is deleted, a local mesh density is applied and the entire part is meshed.

This command can be accessed via several methods including the menu including:

- **Mesh ➡ Define Mesh Density.**
- The Define local mesh densities icon  on the Mesh Manipulation toolbar.
- The Context (right-click) menu from the display area.

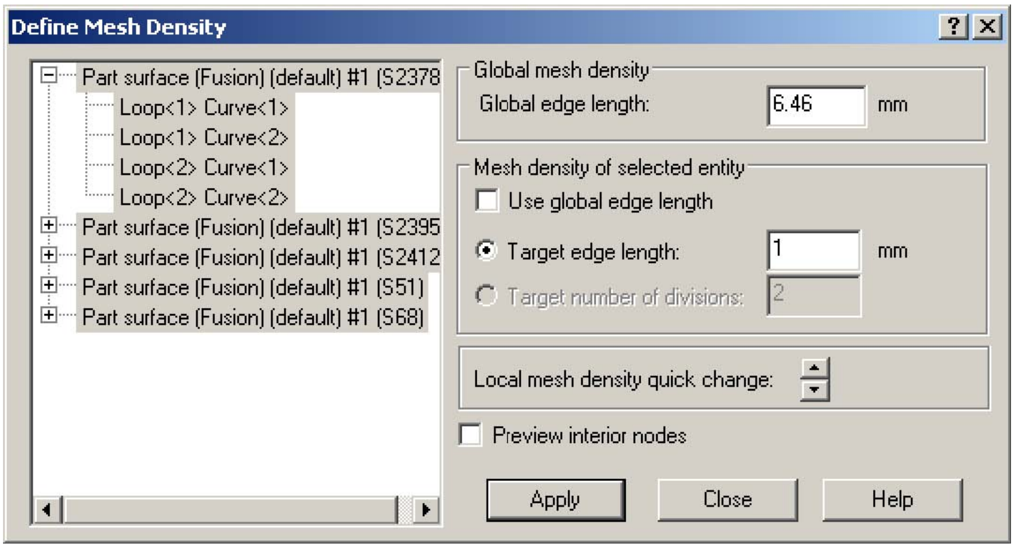


Figure 123: Define Mesh Density

Basic Steps for Changing the Mesh Density

To change the mesh density of a region, follow the steps below:

1. Ensure the surface and/or region layers are turned on.
2. Open the **Define Mesh Density** dialog.
3. Select the features that you want to change the density on.
4. Select all the features in the list.
   - This will set all the features selected to the same density.
5. Uncheck **Use global edge length**.
6. Set the local target edge length.
7. Apply the change.

Figure 124 shows the preview of changing the mesh density on a boss. The local density on the boss is significantly finer than the rest of the part.

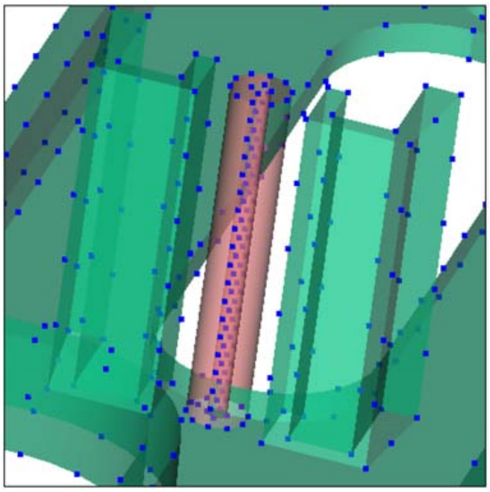


Figure 124: Change mesh density of the part

Once you have selected a surface or region, you can change the mesh density of that whole region; alternatively, you can select loops within that region and change the mesh density of those loops only. Any of the standard selection methods can be used to select entities for mesh density modification. The entities selected are listed in the explorer view on the left side of the dialog. When an individual loop or curve is selected, a blue line on the model shows the location of the curve. See Figure 125.

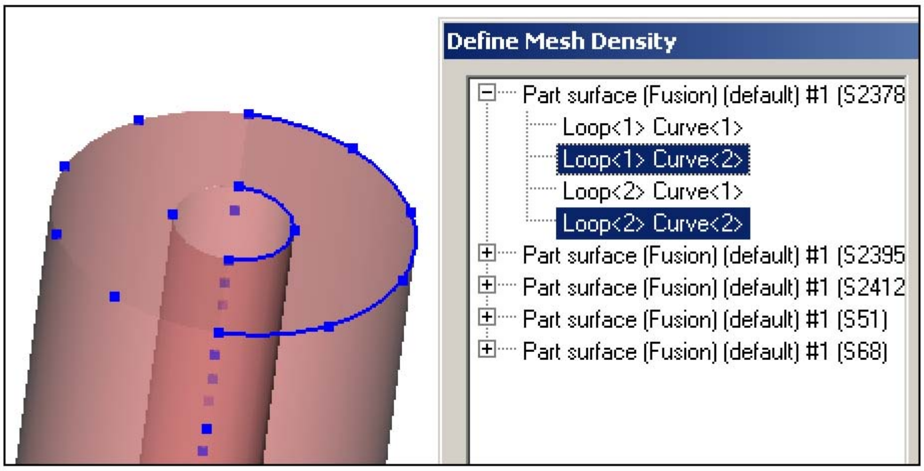


Figure 125: Loops selected, highlighted in blue

From the **Define Mesh Density** dialog, the global mesh density can be changed from the default values. Use the **Apply** button to see the effect of the global edge length change. The blue preview nodes will show the spacing of the nodes when the part is meshed. By default, only the boundary nodes are shown, but the checkbox for displaying interior nodes can be selected. Once you have finalized all the mesh density settings, you can generate the mesh using the standard **Generate Mesh** command. If you want to refine the mesh density further, you can delete the mesh with an undo command, then make the necessary changes to the mesh density and remesh once more.

Fusion and midplane mesh generation settings

Most CAD files require meshing. The Mesh Generator tool shown in Figure 126 is accessed from several places including:

- **Mesh ➡ Generate Mesh.**
- The mesh icon  on the Mesh Manipulation toolbar.
- Double-click the mesh icon  in the study tasks list.

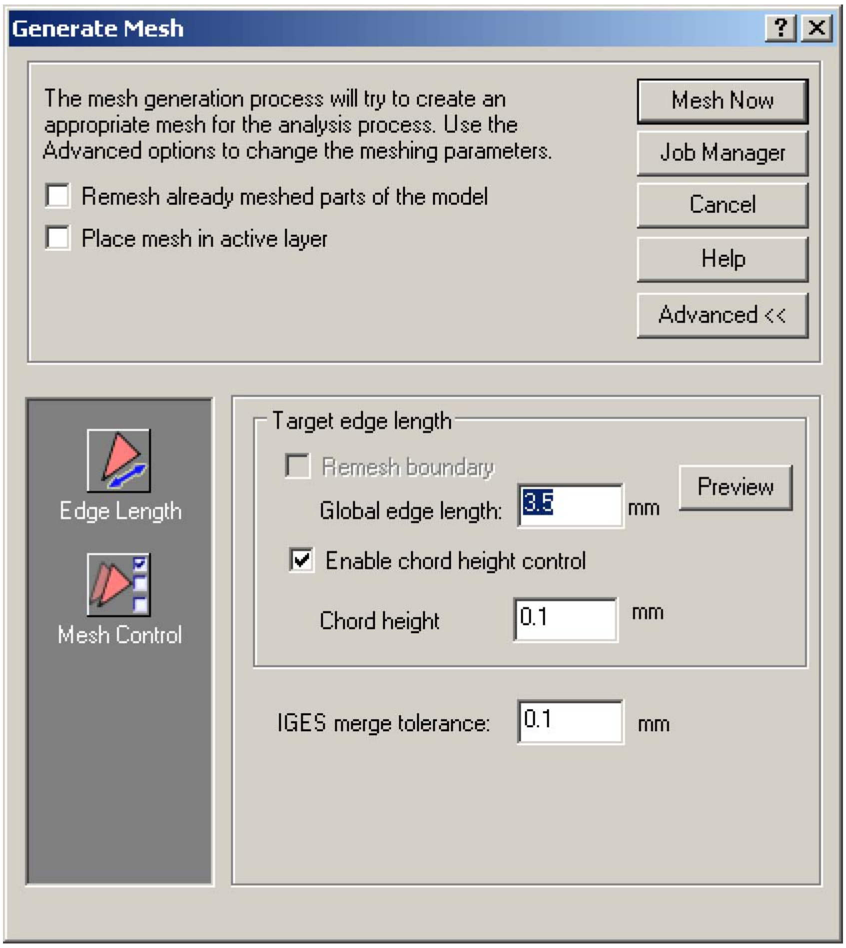


Figure 126: Generate mesh

The Generate Mesh tool is shown in Figure 126. The two check boxes in the top portion include:

- Remesh already meshed parts of the model:
  - This is used when you want to generate a mesh based on an existing mesh rather than the original geometry or STL file. This option is generally not needed. Normally if the first mesh had problems or is the wrong density, it is better to delete the mesh and mesh from the original format. This is used when you want to change the mesh type from Fusion to Midplane for instance.

Place mesh in active layer:

- Normally when meshing, new layers are automatically generated. Activating this box places the new mesh on the active layer rather than creating new layers. This is useful when you are meshing curves for runners or cooling channels, rather than meshing parts.

In Figure 126, the **Advanced** button has been selected, displaying the two advanced options:

- Edge Length.
- Mesh Control.

A description for these advanced options is provided below.

Edge Length

Under Edge Length there are three settings:

- Global edge length.
- Enable chord height control.
- IGES merge tolerance.

Global edge length

This allows you to set the target element edge length when it generates your mesh. This value is the main control for the mesh density of your part. When the default global edge length is changed, the **Preview** button can be clicked to update the preview nodes on the part. This will give you an indication of the mesh density of the part. See Figure 127 for two examples at different densities.

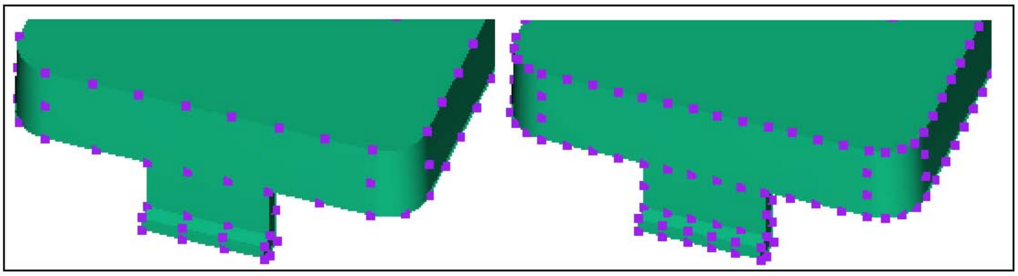
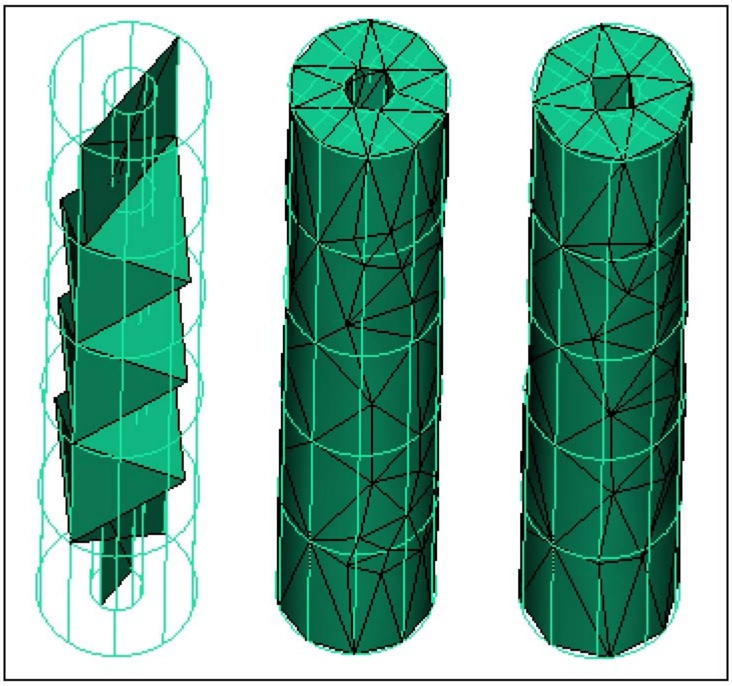


Figure 127: Mesh preview nodes at two mesh densities

Enable chord height control

This check box and text box will ensure that curved features have a mesh on rounded features. The box is checked by default, with a default chord height is 0.1 mm as shown in Figure 126 on page 201. This is a global value only.

To use this feature, check the box then preview the model. Look in areas with smaller radii to see if the mesh around the feature is good or not. Increase or decrease the size as needed. Figure 128 shows how the use of chord height control influences the mesh on a small post on a part. Without the chord height, the global edge length creates a mesh that is collapsed. With the chord height on, the setting influences the mesh density around the part.



Chord height off | **Chord height 0.1 mm** | **Chord height 0.2 mm**

Figure 128: Chord height control examples

IGES merge tolerance

This specifies the minimum distance between nodes. If nodes are located closer together than the specified merge tolerance, they will be merged. Normally the default value is fine, however, it the part is very small and there is a chord height below 0.1 mm, the tolerance may need to be lowered.

Mesh Control

On the mesh control page of the Generate mesh tool, shown in Figure 129 there are five options including:

- NURBS Surface Mesher.
- Optimize aspect ratio by surface curvature control.

- Optimize aspect ratio by proximity control.
- Match mesh.
- Smooth mesh (NURBS Surfaces only).

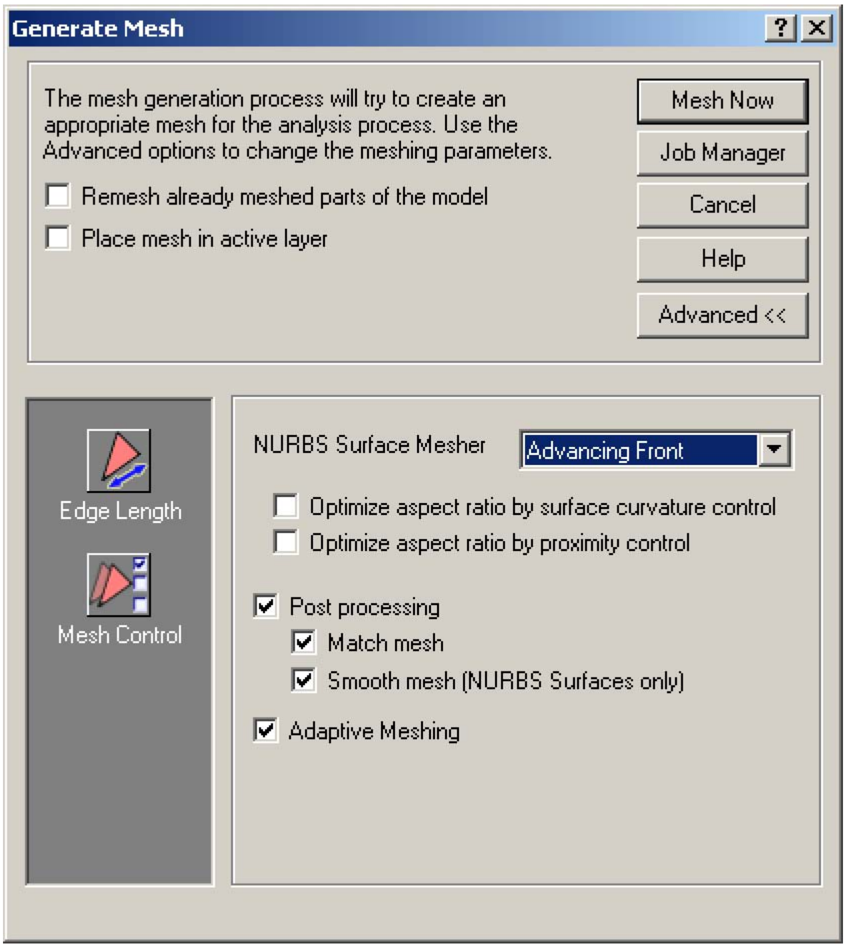


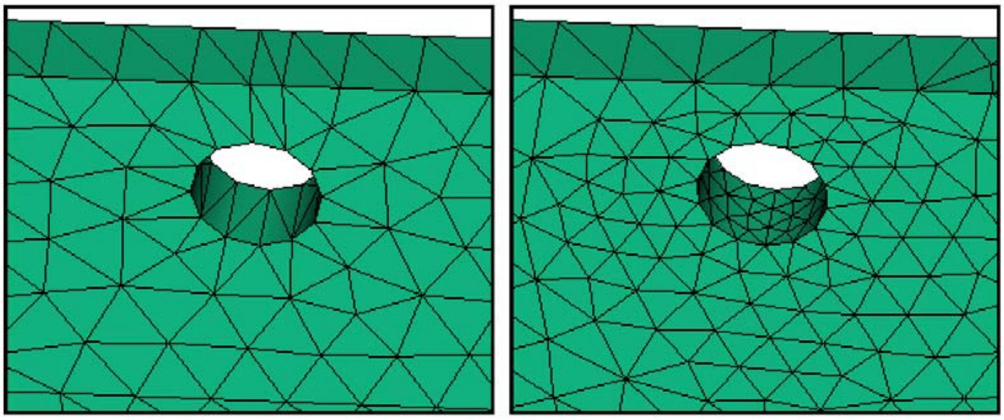
Figure 129: Mesh control page of the Generate Mesh tool

NURBS Surface Mesher

The default mesher is the Advancing Front algorithm. First introduced in MPI 5.1 it is now the default. The second mesher, is now called Legacy. This is the split-based algorithm. The Advancing Front algorithm should be used as the default mesher.

Optimize aspect ratio by surface curvature control

This control affects the meshing of curved surfaces. Surfaces are meshed to maintain the curve, a low aspect ratio and the specified chord height, by using a finer mesh than the global value. The mesh will then be graded out to the nominal mesh size. Figure 130 shows an example of curvature control. Notice how the mesh density is finer both around and through the hole. Curvature control is off by default. To use this option, the chord height control must be on.

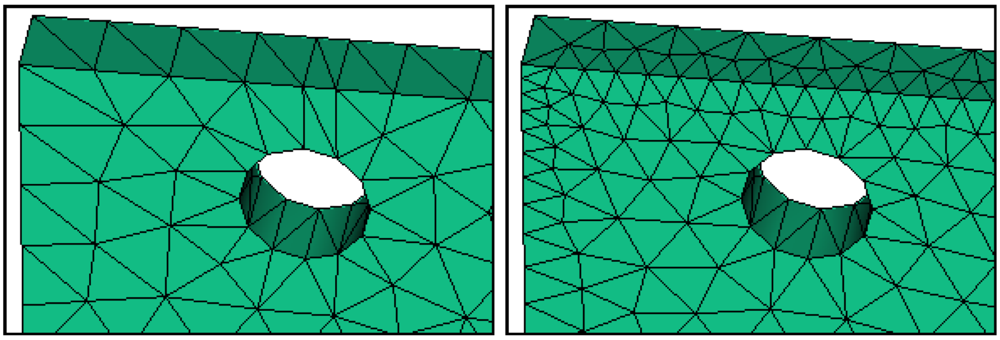


No curvature control With curvature control

Figure 130: Curvature control

Optimize aspect ratio by proximity control

This control reduces the mesh size in areas where the distance between two edges is less than the global edge length. The mesh will be graded out to the nominal mesh density. Figure 131 shows an example of proximity control. The wall thickness of this feature is less than the global edge length so the edge is meshed finer than the global edge length. Proximity control is off by default.



No proximity control With proximity control

Figure 131: Proximity control

Both curvature control and proximity control must have **Chord height** on to work.

Match mesh

This control modifies the mesh after its initial creation to get a better mesh match between elements. Figure 132 shows the same area of a part with and without the mesh matching. The nodes are lined up on the matched part. A better mesh match increases thickness accuracy and leads to more accurate Fusion analyses in general. Mesh matching does not apply to Midplane models. For 3D, the conversion of surface elements to tetrahedral elements is based on mesh matching but is not a primary contributor.

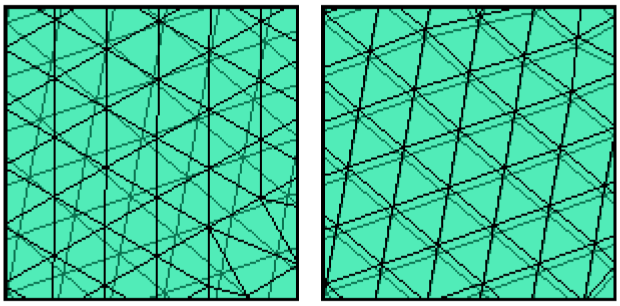


Figure 132: Mesh matching lines up nodes

Smooth mesh (NURBS Surfaces only)

This option shifts the position of nodes around to produce a lower average aspect ratio. This option is on by default, and should be left on.

Surface mesh guidelines

Not all of these mesh control features should be activated on every model. There is always a trade-off between computation time and the number of elements. Generally, results are better with more elements, but if the computation time is very long, this limits the speed of iterations that solvers can achieve. If you need a particularly fine mesh, you can make a coarsely-meshed model for preliminary work, and a fine mesh of the same model to validate the work.

Global edge length

This is the primary setting for the mesh density of the part. A default value is calculated based on model size. Often this number is too large. Click the **Preview** button to see the relative mesh density on the part. There should be enough elements to capture thickness changes in the model and other fine details. The edge length for Midplane and Fusion can be several times the thickness, but for 3D models, the global edge length should be no more than twice the wall thickness, particularly in thin areas.

Chord height

Chord height, the maximum error in distance allowed when converting a curved surface into the planar surfaces that it becomes in a mesh, is on by default, with a default value of 0.1 mm. A smaller chord height value makes a finer mesh that more closely matches a curve. Use the **Preview** button and look at curved features on the part to determine if the value is correct. There should be at least two facets (preferably three or more) around curved features.

Match mesh

Sometimes, activating mesh matching will create thin high-aspect-ratio elements called slivers. Although the match ratio will be much higher, there may be some slivers that need to be cleaned up.

For Midplane models, and for Fusion models that will be converted to 3D, having this option off will be best. For Fusion, most of the time the option should be on. If there are many sliver elements, re-mesh the part with the option off and compare match ratios. If the match ratio is nearly as high, use the mesh created with the option turned off.

Smooth mesh

Smooth mesh should always be on.

Surface curvature control

Curvature control attempts to make equilateral triangles on curved surfaces to maintain the specified chord height This feature is most useful for highly-curved components that will be converted from Fusion to 3D meshes. This feature will produce many elements and so might not be suitable for all Fusion models. It may produce a mesh that is too fine, and may reduce the match ratio. If the geometry has small features that are not needed for Fusion (for example, small fillets) this feature will mesh them, significantly increasing the number of elements with little benefit to the model. Try meshing the Fusion model with and without this control on to see how it meshes the part.

Proximity control

Proximity control meshes areas of parts where edges are closer together than the global edge length at a finer density. This may be useful for 3D parts. However, for Fusion and midplane models, this may create more elements than is desired. Consider meshing the part with and without this control on to see the influence on the part.

3D Meshing

To create a 3D mesh in MPI, you first need to import, mesh and clean up a part as a Fusion model (as described above), then remesh it for 3D. Refer to Figure 116 on page 189. Review the **Model Requirements** chapter to see what attributes a Fusion model must have before being converted. Later in this chapter you will learn how to clean up a Fusion model.

Generate a 3D Mesh

To generate a 3D mesh from a Fusion mesh, the following steps are shown in Figure 133 and are:

- Set the mesh type from Fusion to 3D.
- Set the 3D specific mesh options including:
  - Minimum number of elements through the thickness.
  - Use surface mesh optimization.
  - Use surface mesh matching.
  - Tetra aspect ratio control.
  - Node biasing through thickness.
- Generate the mesh.
- Check and fix the mesh.
- Confirm the mesh is fix and is correct.

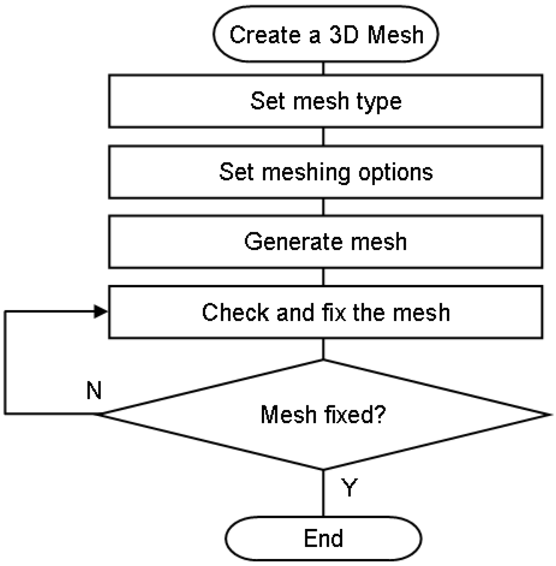


Figure 133: Create a 3D mesh flow chart

Set the mesh type

The mesh type must be switched from Fusion to 3D. This will allow the mesh generator to create tetrahedral elements rather than triangular elements. Setting the mesh type is done from the Study Tasks list. Right click on the Fusion mesh icon and select **Set mesh type ➡ 3D**.

Set the 3D specific mesh options

The Mesh Generator tool shown in Figure 134 is accessed the same way it is for Fusion models, including:

- **Mesh ➡ Generate Mesh.**
- The mesh icon on the Mesh Manipulation toolbar.
- Double-click the mesh icon in the study tasks list.

Most of the tetra mesh options can be left at their default values. The option typically changed is the number of elements through the thickness, if any are changed.

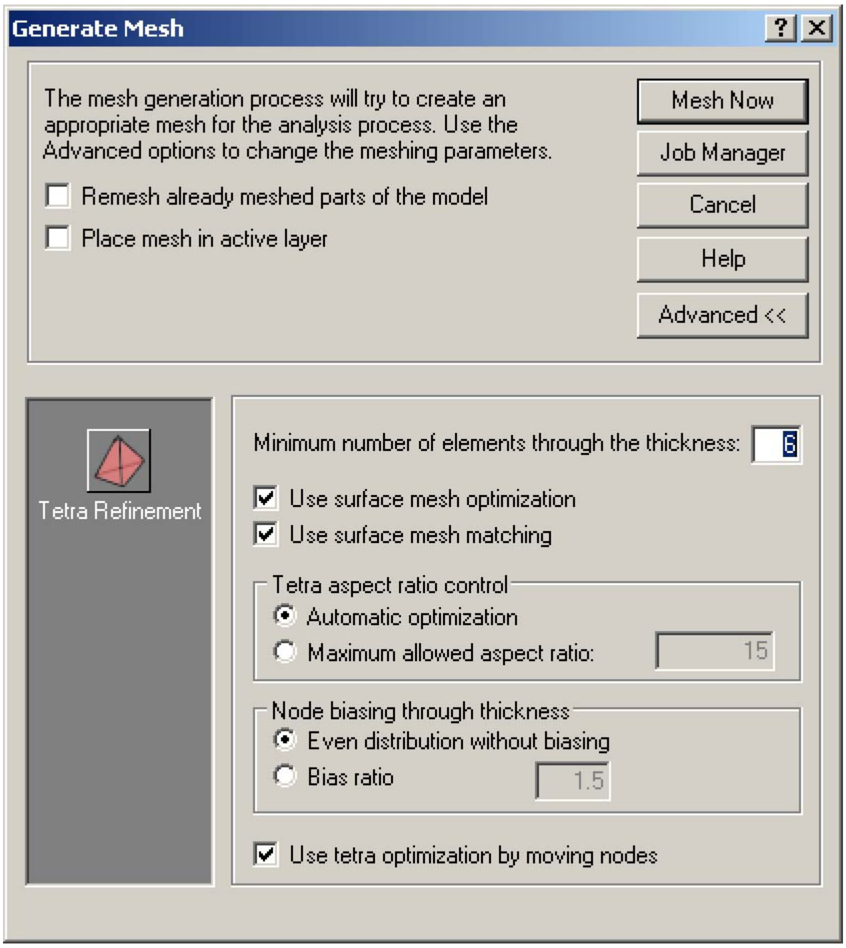


Figure 134: Mesh Generator for a 3D mesh

Minimum number of elements through the thickness

The default number of layers through the thickness is 6. This is fine in most cases. If a fiber flow analysis is run in preparation for warpage, Moldflow recommends 8 layers. Preliminary work can be done with as many as 4 layers. If the analysis needs to pick up very fine detail such as temperature imbalances that might occur in runner systems, then 10 or more layers are needed. To keep the aspect ratio down, and the minimum included angle up, a finer surface mesh is needed. This significantly increases the number of elements.

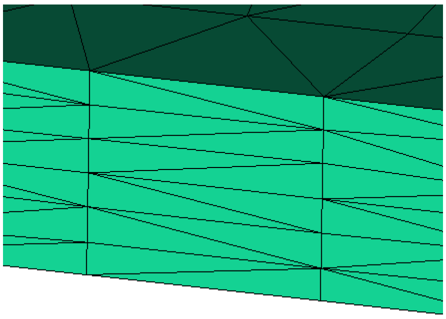


Figure 135: Layers through the thickness of a 3D mesh

Use surface mesh optimization

Tetrahedral (3D) mesh generation starts from triangular surface mesh; therefore, the mesh quality of the surface is very important. Moldflow recommends that you check the surface mesh for:

- Overlaps.
- Intersections.
- Free edges.
- High aspect ratio elements.

If you find any problems, you should fix them manually before launching the 3D mesh generator.

Moldflow recommends that the surface mesh be good before converting. Just in case there are problems that were overlooked or ignored, the option **Use surface mesh optimization** is available. This option can help reduce some meshing problems, especially high aspect ratio elements. When the **Use surface mesh optimization** option is enabled, nodes that are too close together will be automatically merged, eliminating high aspect ratio elements. Figure 136 shows an example of high aspect ratio elements that would be eliminated by merging nodes, in order to improve the resulting 3D mesh.

By default, surface mesh optimization is enabled and should remain enabled in most cases. If the surface mesh is worse after optimization, remesh the part with this option turned off.

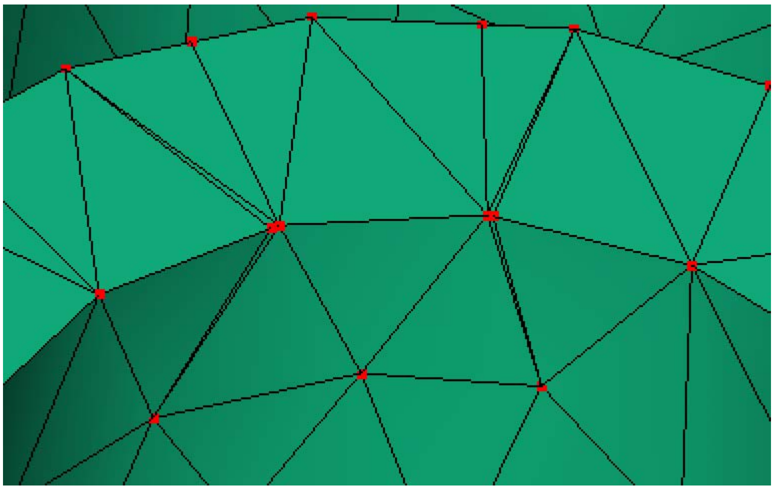


Figure 136: Surface mesh with sliver triangles

Use surface mesh matching

One of the major steps during 3D meshing is to create internal nodes through the part thickness. By default, the mesh generator searches for a matched opposite element for each triangle, and determines the thickness direction by the matched elements. However, not all models have a majority of matched elements. For example, in Figure 137 below, most of the triangles are not matched. In general, Moldflow recommends that you enable the **Use surface mesh matching** option, which is the default. However, if you discover after the part is meshed, the surface of the mesh has changed significantly, you should remesh the part with this option turned off. The surface mesh changes usually with chunky parts, or parts with Fusion meshes that do not have many matched elements.

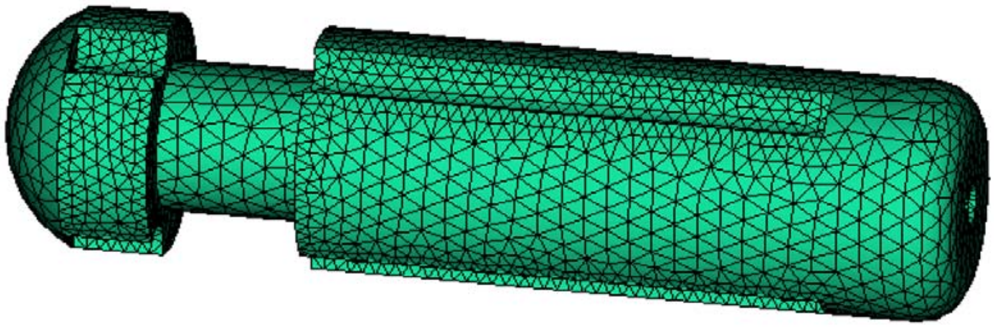


Figure 137: Part with low surface mesh matching

Tetra aspect ratio control

Automatic optimization

When this setting is enabled, optimization of the mesh occurs automatically as soon as the mesh is first created. After optimization, nodes are moved to reduce the aspect ratio of the elements. Since optimizing the mesh and reducing the aspect ratio are a decoupled process, the total number of elements is reduced and so is the average and maximum aspect ratio.

Automatic optimization is the default and produces a good mesh the vast majority of the time.

Maximum allowed aspect ratio

When the maximum allowable aspect ratio is manually set, the element orientation and aspect ratio algorithms are combined. You should use this method if the automatic optimization results in unusually high aspect ratios.

Node biasing through thickness

Biasing is the ratio of the thickness of an inner layer vs. the outer layer. A ratio higher than one for example, 1.2, means that nodes will be closer to the surface of a part's thickness. A bias ratio lower than one for example 0.8, means that nodes will be closer to the center of a part's thickness. Enter a number between 0.5 and 2. Figure 138 shows the effect of biasing. A bias higher than one is generally done to capture shear rate and shear heating better. A bias less greater than 1 is normally done to capture gas penetration or jetting better.

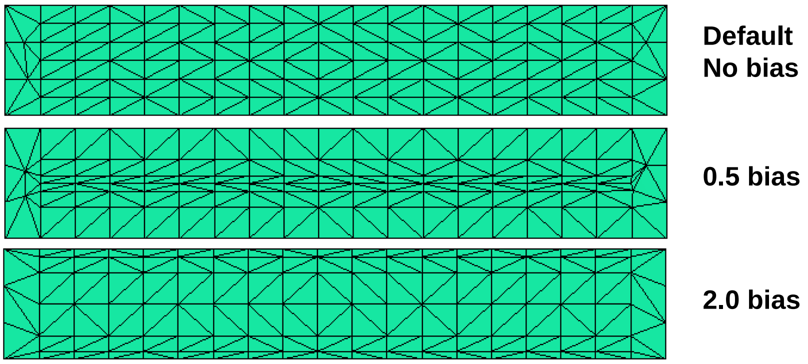


Figure 138: Bias examples

Generate the mesh

Generating a mesh for 3D is the same as it is for Fusion. This subject is discussed in the topic **Generate the mesh** on page 213.

Check and fix the mesh

Fixing a 3D mesh is done automatically with the Mesh Repair Wizard. This subject is discussed in the topic **Mesh Repair Wizard for 3D meshes** on page 228.

Confirm the mesh is fix and is correct

Confirmation for fixing the 3D mesh is by running the Mesh Repair Wizard a second time. Possibly in fixing some problems, new problems can be created in a category already checked. This occurrence is rare but can happen. Using the Wizard twice will confirm the fixing.

3D Meshing Guidelines

Moldflow suggests that you adhere to the following 3D meshing guidelines:

1. Use an appropriate edge length to generate the surface mesh. For thin models, the recommended edge length can be up to twice the thickness.
    - If the edge length is too large, it will be difficult to keep the aspect ratio low and the minimum included angle greater than the recommended 2 degrees.
2. When importing geometry, initially import the model as Fusion, clean it up to the recommendations summarized in the topic **Mesh requirements summary** on page 176.
3. If generating a 3D mesh fails, manually fix the local surface mesh. The mesh log will indicate the area it has trouble with.
4. Run the Mesh Repair Wizard after generating the mesh Ensure the mesh is within the default specifications used in the Wizard.

Midplane mesh generation

To generate a 3D mesh from a Fusion mesh, the following steps are shown in Figure 139 and are:

- Set the mesh type from Fusion to midplane.
- Check the box **Remesh already mesh parts of the model** on the Generate mesh tool.
- Generate the mesh.
    - This will run the Midplane generator and collapse the Fusion mesh into a midplane mesh. A license is required for the Midplane generator.
- Check and fix the mesh.
    - Generally, the there is some cleanup to do on the model.

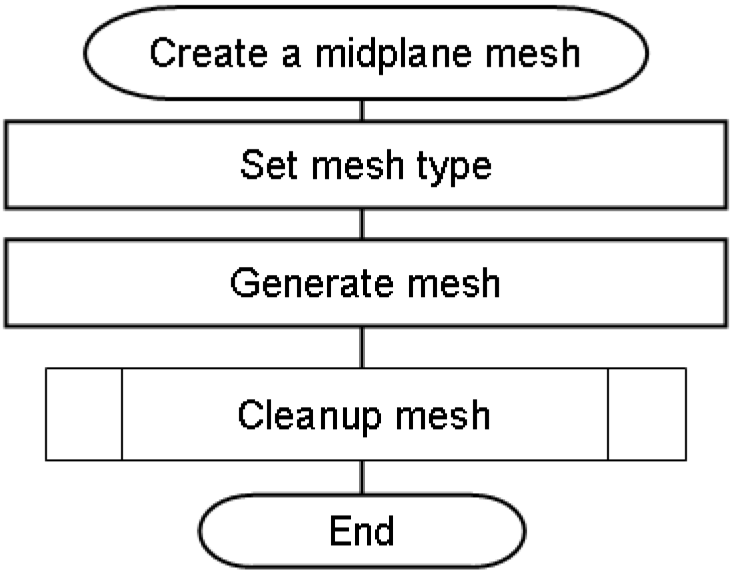


Figure 139: Generate a midplane mesh flow chart

Generate the mesh

To generate the mesh all that needs to be done is click the **Mesh Now** button on the Generate mesh tool as shown in **Figure 134 on page 209**. This is assuming all the mesh options have been set. However, there is another option. By clicking the **Job Manager** button the meshing of the part can be sent to any analysis queue.

Meshing is controlled by the job manager. Even if the **Mesh Now** button is clicked, the mesh is sent to the Priority queue in the Job Manager. This allows you to use Synergy for other things while the part is being meshed. The vast majority of the time the part is meshed by clicking the Mesh Now button.

Evaluate the mesh

Once the geometry has been meshed, the mesh has to be checked and compared to the mesh requirements discussed in the chapter **Model Requirements** on page 163.

Visual inspection

The first step is to visually look at the mesh. This is the best way to determine if there is an adequate mesh density on the part to capture thickness changes. It also is an easy way to get a sense if the mesh density is about right or not. In Figure 140, the same part is meshed at 3 significantly different densities. It is difficult to judge the entire part based on this small area, but the mesh on part (A) has only 1004 elements. This mesh is a bit coarse for the part. This is a Fusion model. The match ratio will be a little low and this part may not represent the thickness changes well. Part (B) has 58750 elements, many more elements than is necessary. The main problem here is compute time. Part (C) has about the right number of elements. There are several rows of elements up the side, and the elements are mostly equilateral or close to it. This mesh has 8178 elements. The analysis time for part (A) is under one minute, part (B) is over 4 hours, and part (C) takes under 10 minutes. The pressure prediction between the second and third parts is under 3%. Just looking at these parts, part (B) can be ruled out as too many elements, and by inspecting the whole part it is clear that part (A) is not good either.

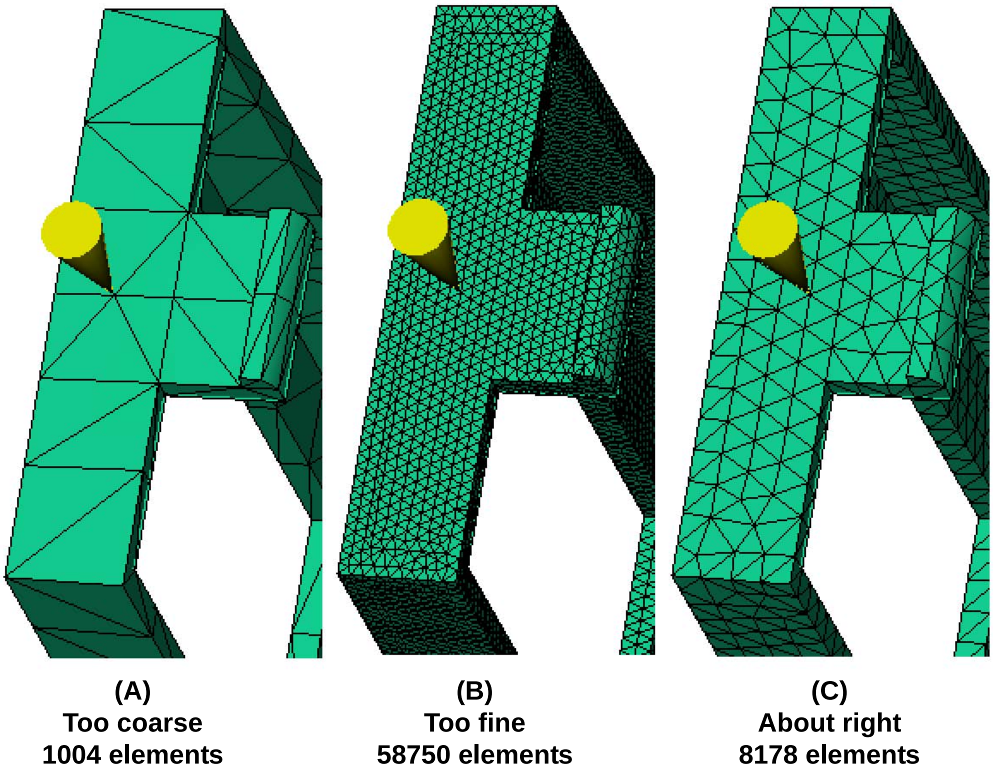


Figure 140: Visual inspection of the part

Mesh Statistics

Once visual inspection has determined the number of elements "looks" about right, use **Mesh Statistics** to start to determine if the mesh is acceptable. To access Mesh Statistics, click the icon  on the Mesh manipulation toolbar or by **Mesh ➡ Mesh Statistics**. The **Mesh Statistics** dialog summarizes the quality of a mesh. It is broken down into the following six sections:

- Entity counts.
- Edge details.
- Orientation details.
- Intersection details.
- Surface triangle aspect ratio.
- Match ratio.

The Mesh Statistics summary, shown in Figure 141, is a good way to check if there are any problems with the mesh, and if so, what the problems are. Once a problem has been identified in the mesh statistics, corrective action can be taken to fix the problem.

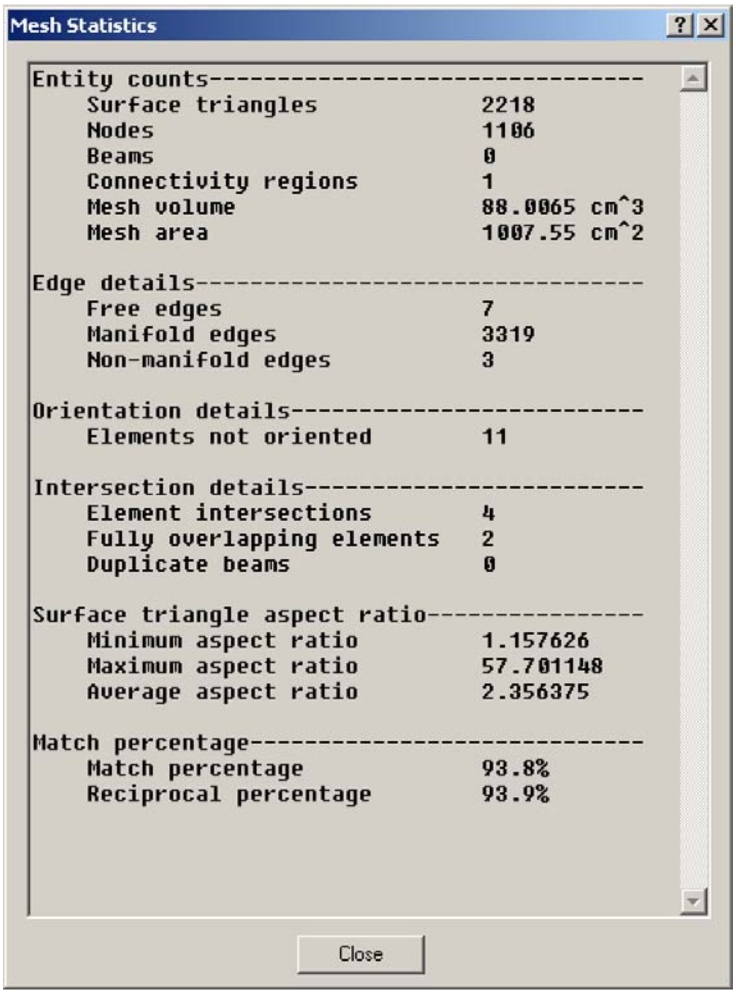


Figure 141: Mesh Statistics

In Figure 141, there are a number of problems indicated by the statistics including:

- Both free edges and non-manifold edges are present.
    - In a good mesh there should be none of these edge types.
- Eleven elements are not oriented.
- Two elements are fully overlapping.
- Four elements are intersecting other elements.
- The maximum aspect ratio is above the limit of 6:1

When faced with a scenario like this, you should now use the diagnostics tools to identify each of these problems in the model so they can be fixed.

Diagnostic tools

The diagnostics show in more detail the problems listed on the Mesh Statistics dialog. Normally, when a statistics report is run and a problem is found, a diagnostic is executed. For instance, if the aspect ratio is too high, the offending elements are displayed together with a diagnostic. Diagnostic tools are found several ways including:

- In the **Mesh** ➡ **Mesh Diagnostics** menu.
- On the Mesh manipulation toolbar.
- On the Tools Pane, in the toolbox, in the Mesh Diagnostics icon .

There are many diagnostics for both filling and cooling applications. They include:

- Aspect Ratio Diagnostic.
- Beam L/D Ratio Diagnostic.
- Overlapping Elements Diagnostic.
    - This includes intersecting and overlapping elements.
- Orientation Diagnostic.
- Connectivity Diagnostic.
- Free Edges Diagnostic.
    - This includes free and non-manifold edges.
- Thickness Diagnostic.
- Occurrence Number Diagnostic.
- Zero Area Elements Diagnostic.
- Fusion Mesh Match Diagnostic.
- Beam Element Count Diagnostic.
- Trapped Beam Diagnostic.
- Centroid Closeness Diagnostic.
- Cooling Circuit Diagnostic.
- Bubbler/Baffle Diagnostic.

Diagnostic pane

The diagnostics are located on the Tools pane, as shown in Figure 142. When one is activated by any of the methods described above, the Tools pane opens, if necessary, and all the diagnostics are listed in the combo box below the toolbox. The first eleven diagnostics have a shortcut key F2-F12 as another method of navigating between the diagnostics.

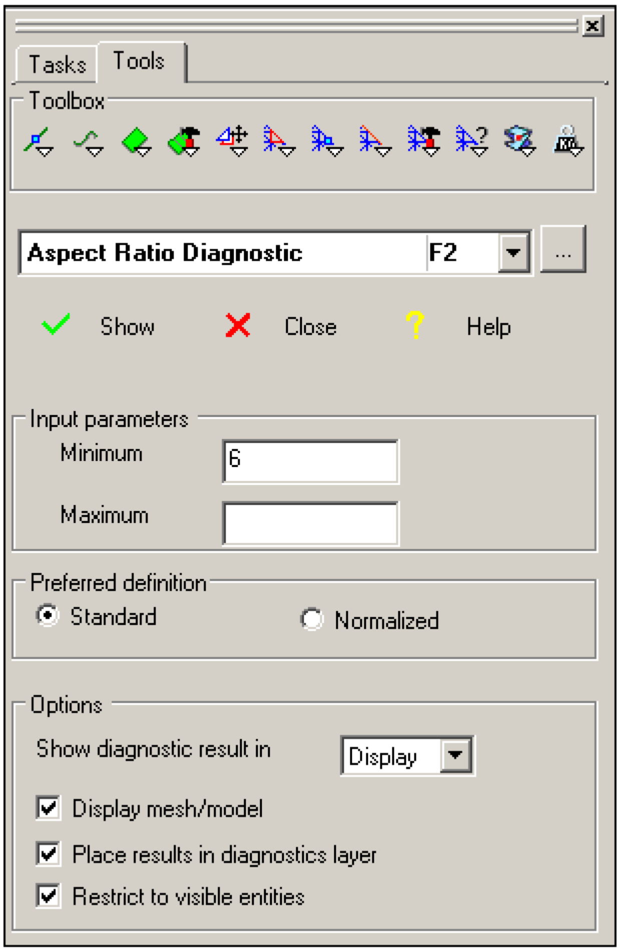


Figure 142: Aspect ratio diagnostic

Diagnostic input

Depending on the diagnostic, there may be some input parameters that are necessary. The aspect ratio diagnostic shown in Figure 142 is one of the most commonly used diagnostics. The input here is the minimum aspect ratio it will display. The default is 6. When the diagnostic is shown, it will display only elements that have an aspect ratio higher than 6:1. Some diagnostics have no input, others have checkboxes for indicating what will be displayed, such as displaying either intersections and overlaps.

Diagnostic options

The options for diagnostics include the following items:

- **Show diagnostic result in**:
  - ➤ **Display** - This is the default option. This will produce a graphic display of any entity for the diagnostic. If there are no elements to be displayed, the diagnostic will not appear.
  - ➤ **Text** - This will be a similar output to the mesh statistics but will only show results for the single diagnostic.
- **Display Mesh/model**.
  - ➤ This is checked by default. It will display mesh in addition to the diagnostic result.
- **Place results in diagnostics layer**.
  - ➤ By default this is unchecked. When checked, any entities that are displayed will be placed on a diagnostics layer that will be created. This is useful for diagnostics that have tolerances or input parameters such as Aspect ratio. This separates the problem entities from the rest of the model. This aids in fixing the problems.
- **Restrict to visible entities**.
  - ➤ By default, this is unchecked. When checked, the diagnostic will refresh only for layers/entities that are visible. When fixing the mesh, each time a fix is applied, the diagnostic is re-run to refresh the display. This will take a considerable time when the model is large. Used in conjunction with the Diagnostics layer, it can save significant time when cleaning up large models.

Diagnostic displays

When the show button is clicked, the status bar of Synergy at the bottom left corner of the Synergy window, will display a message indicating the number of entities displayed, as shown in Figure 143.

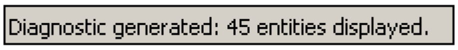


Figure 143: Number of entities displayed

When diagnostics results is displayed as text, a dialog opens similar to mesh statistics. The information will summarize the specific diagnostic only.

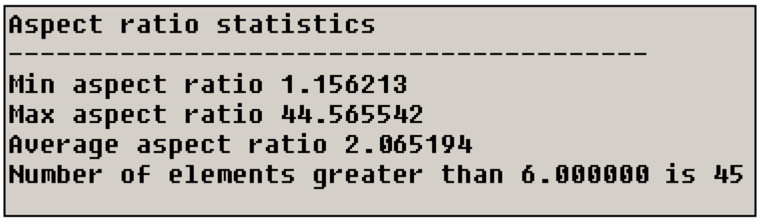


Figure 144: Diagnostics output as text for aspect ratio

Most of the time, the diagnostics are displayed graphically. The aspect ratio diagnostic is shown in Figure 145. Most diagnostics fill in the element according to the value for that element and the scale. Aspect ratio is unique because the diagnostic displays a line that is normal to the element. The color and length of the line correspond to the scale.

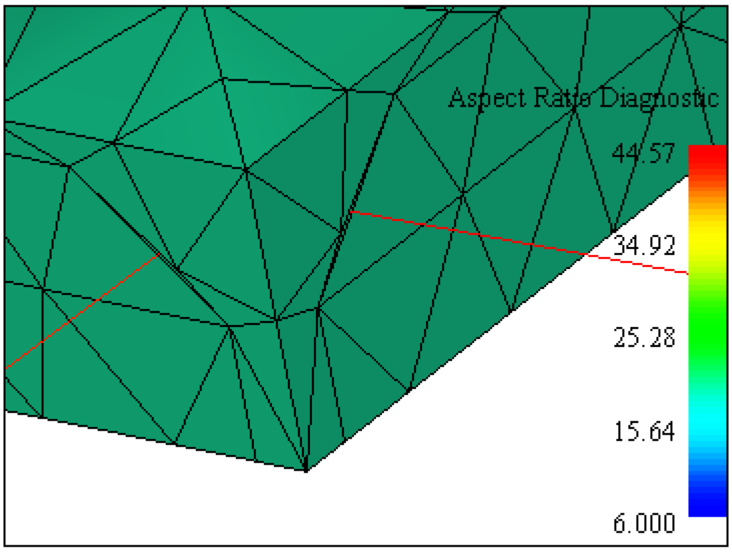


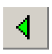
Figure 145: Aspect ratio display

Diagnostic Navigator

When a diagnostic is displayed, the **Diagnostic Navigator** toolbar is automatically displayed, as shown in Table 146. This allows you to quickly zoom on elements in the model that were detected by the diagnostic. The navigator sorts elements by severity, from highest to lowest value. Using the four navigation buttons, you can browse the problem areas and repair them. The navigator automatically centers and magnifies the entity. You can rotate the model to find the best viewing angle. The four navigation buttons are described in Table 23.

Figure 146: Diagnostics Navigator

Table 23: Diagnostic navigator commands

Icon	Command
	First Diagnostic (Highest value)
	Previous Diagnostic
	Next Diagnostic
	Last Diagnostic (Lowest value)

Thickness

In addition to diagnostics looking at mesh problems such as aspect ratio, the diagnostics allow you to create a plot of thickness. Thickness is a common diagnostic to use.

Depending on the file format used to import the model, and the mesh density of the part, the thickness may not be represented exactly as it should be. Generally, the lower the edge length of an element, the better the thickness representation will be. The thickness plot will allow you to see what thickness are used by the solvers. In Figure 147, the query command was used to see the thickness of 3 regions on the part. The **CTRL** key was depressed so that more than one label at a time could be displayed.

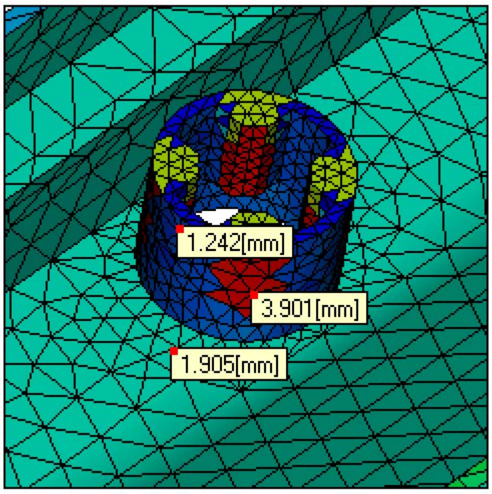


Figure 147: Thickness of a Fusion Model

If the thickness of a portion of the model is not correct, the element thickness can be modified by the element properties, as shown in Figure 148. By default, the thickness of a Fusion element is set to **Auto-determine**. In addition to the thickness diagnostic, the name of the part also has the thickness determined by Synergy. If this thickness is not quite accurate, you can manually change the thickness of the elements. If you change the thickness, make sure you change both sides of the wall thickness. Edge elements as defined by the match ratio diagnostic, are 75% of the wall thickness of the matched elements the edge elements is attached to.

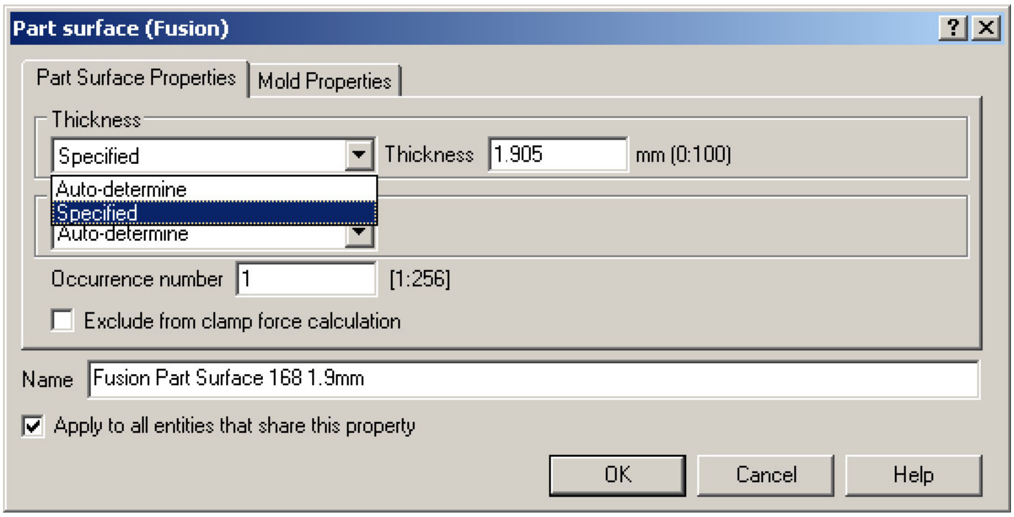


Figure 148: Fusion part properties

Do NOT change the thickness significantly if the model will be used for Cooling and Warpage. Small changes to average out the thickness are fine, but do not use this feature to globally change the wall thickness - there will be errors in the cooling and warpage solvers if you do.

Cleanup the mesh

After the part has meshed, and evaluated using visual inspection, mesh statistics, and diagnostics, problems in the model need to be fixed, so a clean model is used for analysis, as shown in Figure 149. The first step is to use a Mesh Repair Wizard then if any remaining issues remain, a manual cleanup process is used.

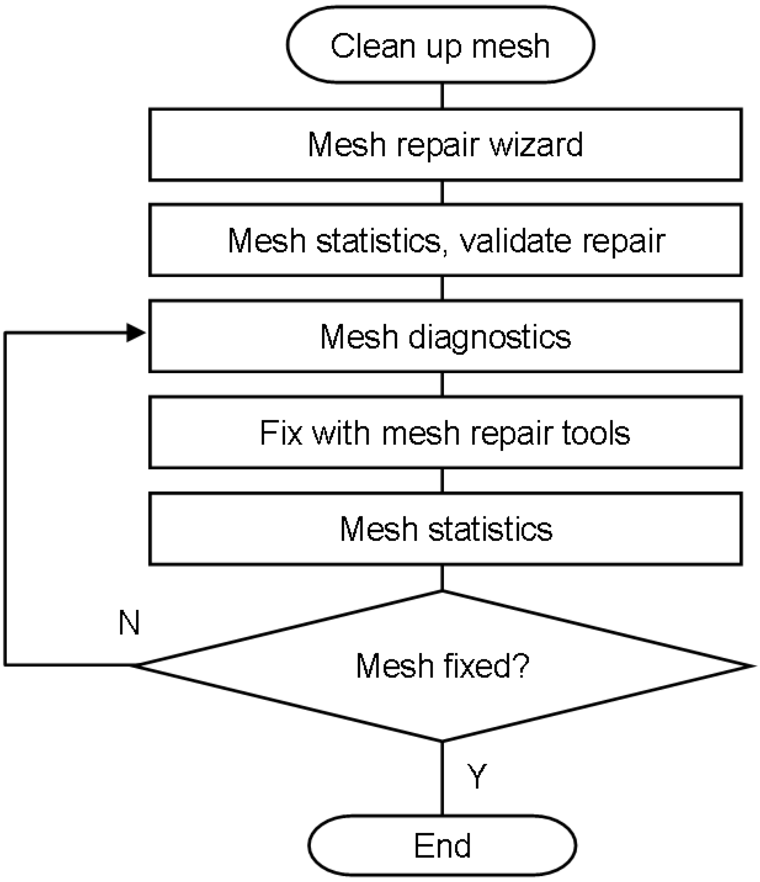


Figure 149: Clean up mesh flow chart

Mesh Repair Wizard

The **Mesh Repair Wizard,** shown in Figure 150, is a tool that automatically cleans up most of the problem areas on your meshed model. The tool is opened with the command **Mesh ➡ Mesh Repair Wizard**, or on the **Mesh Manipulation** toolbar. The Mesh Repair Wizard includes the following pages:

- Stitch Free Edges.
- Fill Hole.
- Overhang.
- Degenerate Elements.
- Flip Normal.
- Fix Overlap.
- Collapsed faces.
- Aspect Ratio.
- Summary.

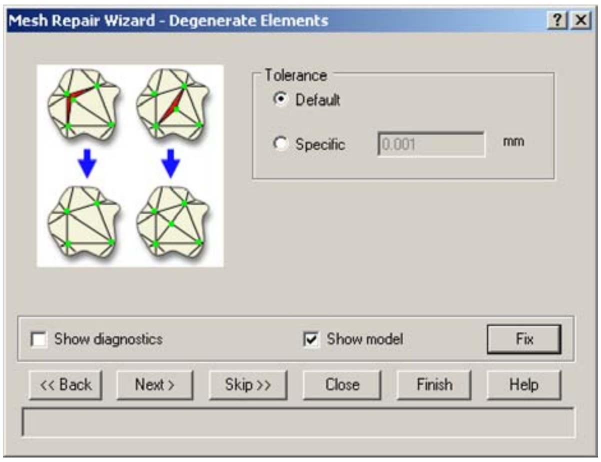


Figure 150: Mesh Repair Wizard, degenerate elements page

Stitch free edges

Stitch Free Edges and **Fill Hole** involve free edges. Free edges are sides of elements that do not touch any other side. This tool looks for areas on the part that have a free edge, and where the distance between nodes is small. The default tolerance is 0.1 mm, but the tolerance value can be changed. Figure 151A shows how a free edge is stitched, and Figure 151B shows the free edges diagnostic of an edge that should be stitched.

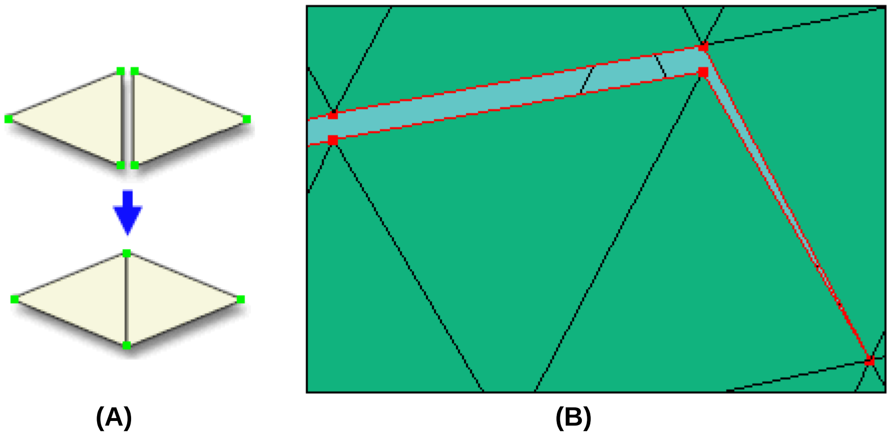


Figure 151: Stitch free edge

Fill hole

Fill Hole involves free edges. The **Fill Hole** page of the Wizard finds nodes on a free edge of a Fusion model, shown in Figure 152 that are above the distance tolerance of the stitch edge command, and then creates elements to fill the hole.

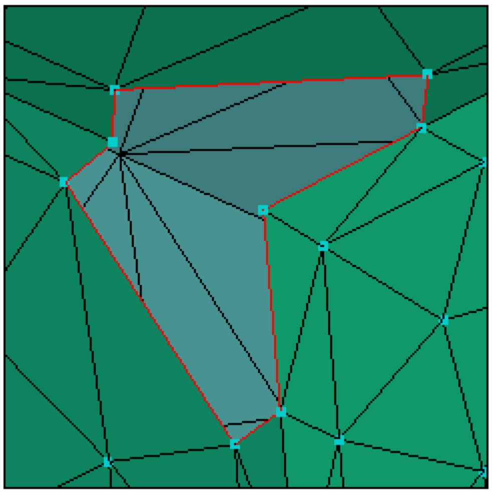


Figure 152: Free edges around a hole a hole

Overhang

An overhanging element is an element that has both non-manifold and free edges. It is an element that should be deleted from the model. Figure 153 shows an example of an overhanging element.

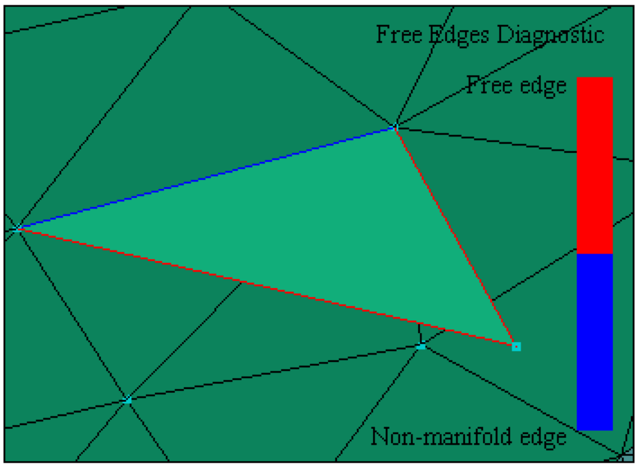


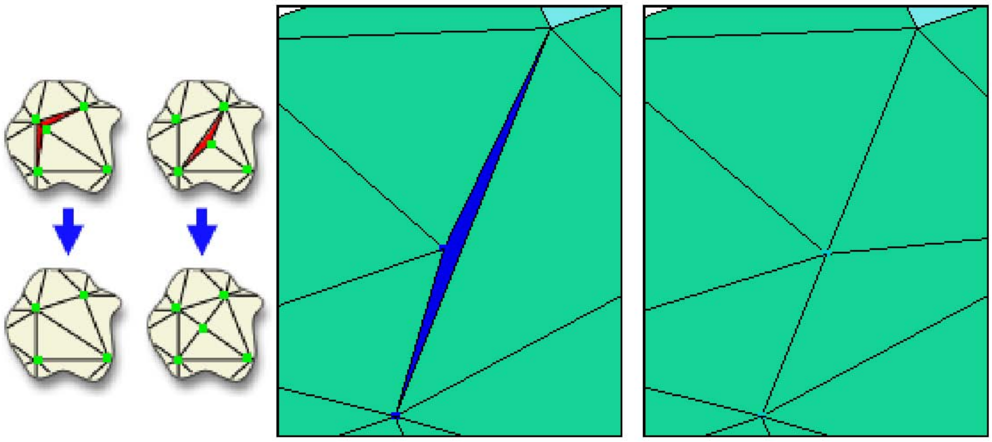
Figure 153: Overhanging element

Degenerate elements

A degenerate element is an element that has a very short height. A degenerate element is defined by an element that has:

- Two or three nodes at about the same location, or
- An element with a node that is very close to the opposite edge.

The default tolerance is very small (0.001 mm). You can, however, specify a different tolerance if you wish. Depending on the type of problem, nodes will be merged or a mid-side node will be created and merged to fix the problem. Figure 154A shows two examples of degenerate elements and how they are fixed. Figure 154B shows an element identified as being degenerate, and Figure 154C shows the mesh after fixing.



(A) Kinds of degenerate elements and how fixed

(B) Degenerate element

(C) Element fixed

Figure 154: Degenerate elements

An efficient use of Degenerate elements is to increase the tolerance. Each time you increase the tolerance, toggle the Show Diagnostics check box on and off. Keep increasing the tolerance until just before important geometry begins to collapse. Use the Diagnostic Navigator to investigate the problem areas. This normally eliminates most high aspect ratio elements in your model.

> Care should be taken with larger tolerances. Be sure to carefully check the model after fixing degenerate elements to make sure the model was not damaged. Use the Undo command if necessary.

Flip normal

The normal of an element defines the top and bottom side of the element. For a Fusion model, the element normals must be consistent. If any problem elements are found, the elements will be flipped as necessary to solve the problem(s).

Fix overlap

The **Fix Overlap** page of the Wizard will identify and fix element overlaps and intersections. An overlap occurs when two elements in the same plane occupy the same area. An intersection occurs when one element passes through the plane of another element. Neither condition should be present in a Midplane, Fusion or 3D model.

Collapsed faces

A collapsed face occurs when a node is used to define elements on opposing faces of the wall thickness. Figure 155 shows an example of this. The wizard will detect the situation and correct the problem by adding a new node in the plane of the problem elements and redefine the elements.

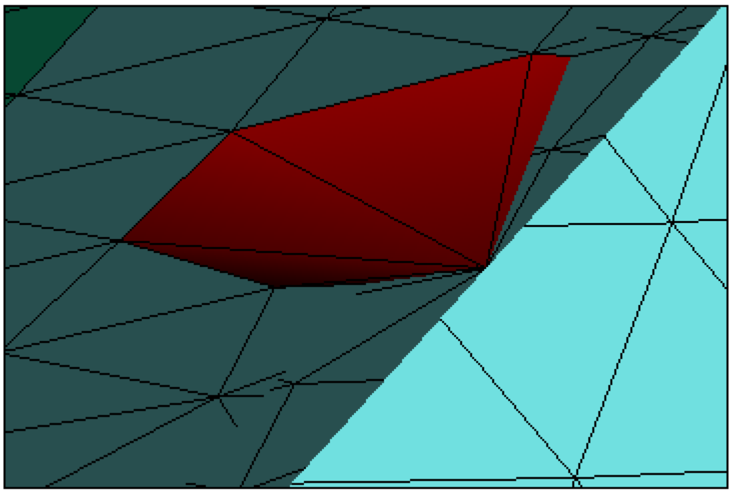


Figure 155: Collapsed face

Aspect ratio

The **Aspect Ratio** page of the Wizard, displays the minimum, maximum and average aspect ratio values. A **Target** aspect ratio box is available, with a default value of 10. You can change the target aspect ratio value as required. When you click the **Fix** or **Next** buttons, the Wizard reduces the element aspect ratios by merging nodes or splitting long elements.

Summary

The final page of the Mesh Repair Wizard shows a summary of the fixes that the Wizard made. It lists all of the problems that were fixed, and how many incidents of each problem were fixed.

Stitch Edges:	0 free edge stitched
Stitch Edges:	0 free edge stitched
Fill Hole:	3 holes fixed
Overhang Fix:	1 overhang element(s) removed
Degenerate Element:	2 elements fixed
Orient All	
Fixed collapsed faces:	1 place(s) modified
Aspect Ratio Fix:	1 elements modified

Figure 156: Mesh Repair Wizard summary

Using the Mesh Repair Wizard

A diagnostic for a given page calculates when you open the Wizard or move to the next page. If a problem is detected, a message appears similar to the message shown in Figure 157. There are two check boxes:

- Show diagnostics.
  - ➤ This will toggle a diagnostic display for the mesh problem found. By default it is deselected.
- Show model.
  - ➤ This will toggle the meshed part. By default it is selected.

Fixing a problem

There are two choices for fixing a problem found on any of the wizard pages, including:

- Next.
- Fix.

The Next button

The Next button will fix the problem from the current page, then go to the next page of the wizard.

The Fix button

The Fix button will fix the problem, and stay on the same page. This allows you to:

- Review the number of changes made.
- Review the model and undo the changes if necessary.
- Change a setting and run the fix again.

If you do not wish to fix any problems on the page, click the **Skip** button.

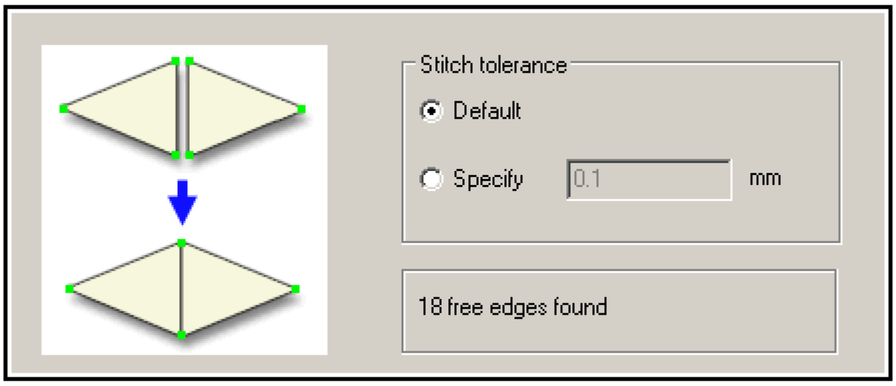


Figure 157: Mesh Repair Wizard - Errors found on Stitch free edges page

Mesh Repair Wizard for 3D meshes

The Mesh Repair Wizard also includes diagnostics and fixes for tetrahedral meshes. Figure 158 shows the first page of the Mesh Repair Wizard. On this page, you can choose which diagnostics to run. By default, all are on.

The diagnostic can also be restricted to entities on visible layers. If the model is very large and the entire model does not need to be checked, entities that do not need checking can be moved to hidden layers and the diagnostics run only on the layers that are visible.

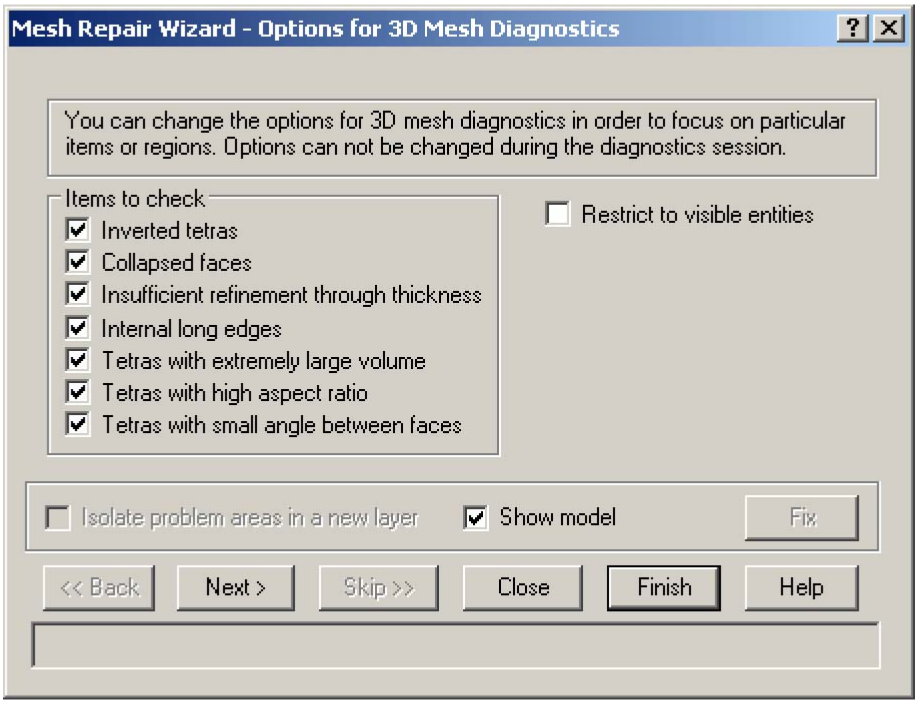


Figure 158: Mesh Repair Wizard, lists problems to check for

The second page of the wizard, shown in Figure 159, displays the results of the diagnostics run and summarizes the following:

- Total number of nodes.
- Total number of tetras.
- Maximum edge length ratio.
- Maximum volume ratio to average volume.
- Maximum aspect ratio.
- Minimum angle between tetra faces.
- Number of collapsed elements on a model boundary.
- Number of inverted elements.
- Status of mesh refinement.

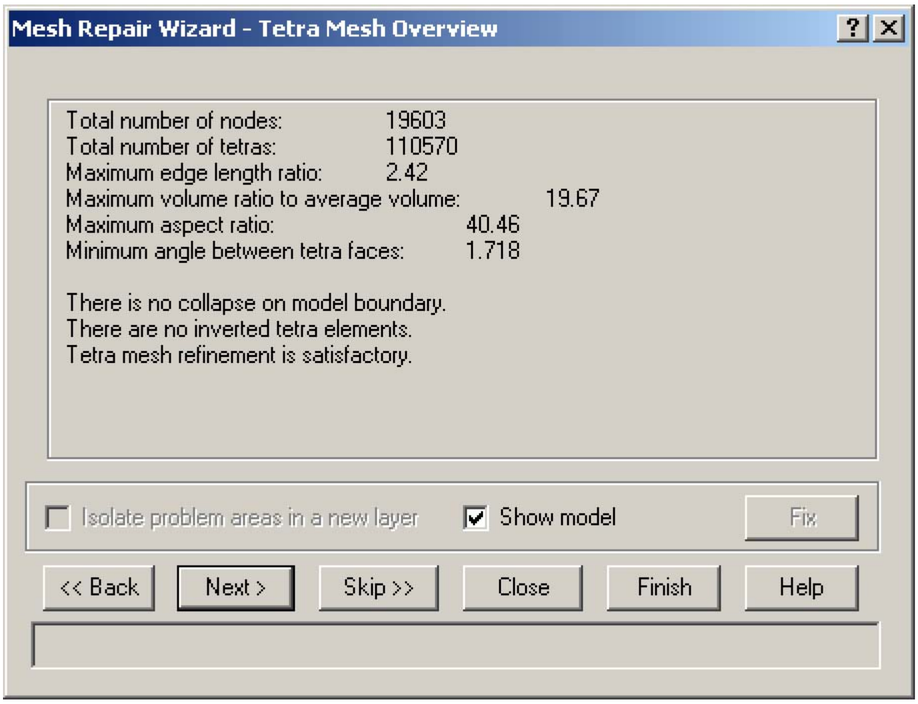


Figure 159: Mesh Repair Wizard, reviews mesh statistics for the tet mesh

Using the Mesh Repair Wizard for 3D meshes

Use of the Mesh Repair Wizard for the tet mesh is very similar to using it for a Fusion or midplane mesh. The two primary ways to fix a problem is with the **Fix** or **Next** buttons.

- Pressing **Fix** repairs the currently-diagnosed problems, and then re-runs the same diagnosis. If further repairs are needed, the **Fix** button can be pressed again until no changes are made or the number of elements in the diagnostic becomes zero. Very often this is the best procedure. How the mesh repair is working can be monitored.
- Pressing **Next** repairs the currently-diagnosed problems and advances to the next diagnosis type, even if there were still some remaining problems.

Several pages of the wizard have tolerances that can be set. To change the tolerances, enter the new value in the field and click update. The default tolerances are good targets. The tolerances can be made tighter to make the mesh even better than the minimums recommended.

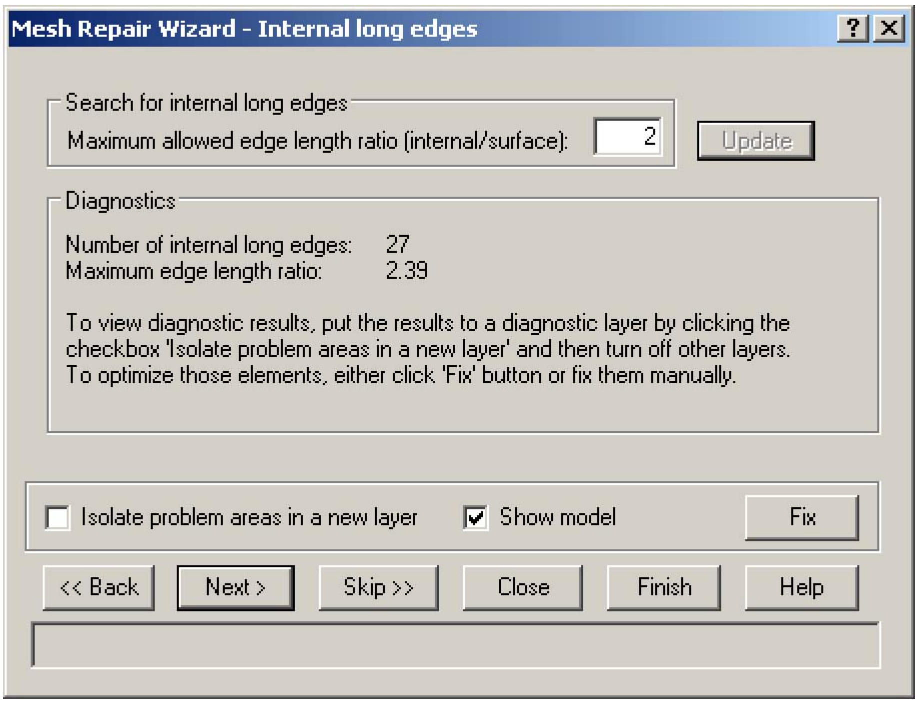


Figure 160: Mesh repair wizard, Internal long edges setting the tolerance

Validate the repairs

Whenever an automatic fix is applied to the mesh, the repairs must be validated to ensure no new problems were created. For Fusion and midplane models, the Mesh Statistics report should be run. For 3D meshes, use the diagnostics of the Mesh Repair Wizard.

Mesh diagnostics

Once the Mesh Repair Wizard has finished and the validation indicates there are still problems, the mesh diagnostics need to be run for the specific problem such as aspect ratio. With specific elements identified that have problems, the mesh repair tools can be used to fix individual problems.

Fix the mesh with mesh repair tools

In many cases, the Mesh Repair Wizard will fix most problems but not all problems. This is practically true for Midplane and Fusion. Manual mesh cleanup must be done to finish the process. The Tools pane contains the toolbox with all the tools in it. The list of tools related to mesh cleanup are listed in Table 24.

Table 24: Toolbox commands for mesh cleanup

Icon	Tool	Commands
	Create/Beam/Tri/Tetra	Create Triangles Create Beams Create Tetras
	Nodal Mesh Tools	Insert Node Move Node Align Node Purge Node Match Node Merge Node
	Edge Mesh Tools	Swap Edge Stitch Edge Fill Hole
	Global Mesh Tools	Remesh Area Smooth Nodes Orient Element Delete Elements Project Mesh Global Merge Auto Repair Fix Aspect Ratio Create Regions Orient All

Navigation between tools

In the process of mesh cleanup, several tools are used often going from one to another frequently. There are many ways to navigate between the mesh cleanup tools including:

- The **Mesh** ➡ **Mesh tools** menu.
- The Toolbox in the Tools pane.
- The mesh manipulation toolbar.
- User defined toolbars.
- F2 - F12 keys.
    - ➤ Once one of the mesh editing tools is opened by other some method, the combo box on the Tools pane gets populated with all the tools, as shown in Figure 161. The first 11 tools have the F keys mapped to them. This is a very easy way to navigate between the tools.

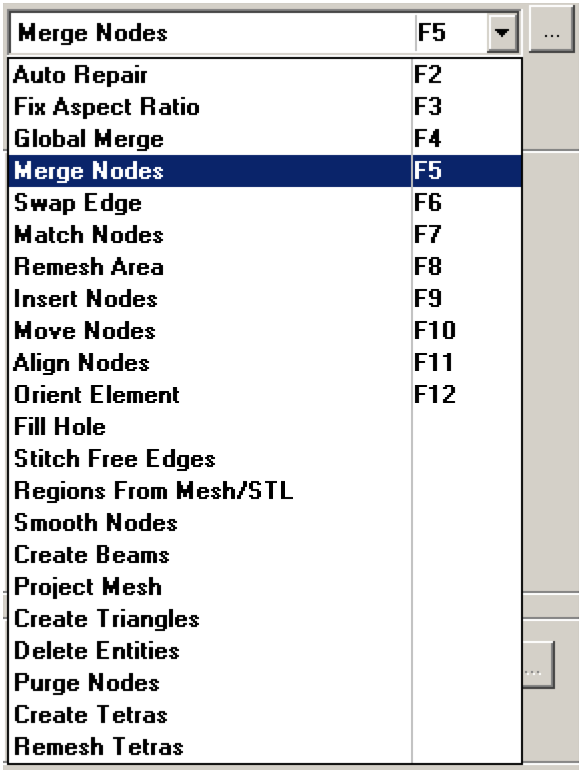


Figure 161: Combo box on the tools pane with mesh cleanup tools listed.

General usage of a mesh repair tool

Most of the repair tools have one or more fields to be filled out. Most require input from the model such as a node or element number. Some fields only require one entity, others support multiple entities. The yellow field is the active field. For fields that require only one entity for input, the active field automatically switched between fields.

Entity filters

For most mesh cleanup tools, entities are selected. A filter is used to determine how entities are selected. For most tools, the choices are shown in Figure 162 and include:

- Any item.
- Nearest node
- Node.

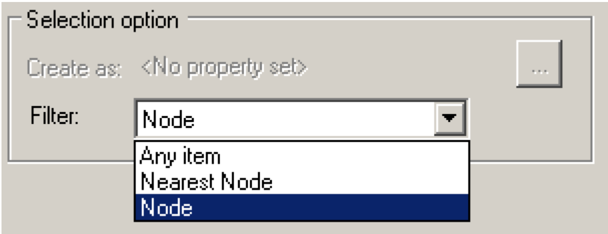


Figure 162: Filters used to select entities

Any item

In the context of the mesh tools, this filter will only allow you to pick nodes.

Node

- This will only allow you to pick nodes.
- If you pick by clicking (rather than band selecting), you must be on the node.
- This filter is the default for most of the mesh tools that require nodes as input.

Nearest node

- This filter will pick the closest node from where the mouse is when you click.
- If you are on or very close to the node, it will NOT select it.
- The nearest node filter can be very handy when a lot of node picking is necessary as you don’t need to be directly over the nodes.

You can switch between the filters at any time. However, when you change to a different tool, the default filter will always be the active filter.

When using mesh cleanup tools, selecting multiple entities is done like it is for anything else, hold down the control key.

The mouse apply can be programmed into the mouse this will significantly speed up the mesh cleanup process. A good definition is to tie the Mouse Apply action to the CTRL + Right mouse. Because the CTRL key is used to select multiple items, one hand can be on the mouse and the other holding down the CTRL key. With one click of the mouse you can go from selecting entities to applying the tool.

Auto repair

Use Auto Repair when there are intersections and overlaps in the model. It tries to fix the problems automatically. There is nothing to enter, just hit the **Apply** button to start the process. On a large model, it may take some time to finish. When done, a report will indicate how many intersections were fixed. This tool has been largely replaced by the Mesh Repair Wizard.

Fix aspect ratio

The fix aspect ratio command tries to automatically fix aspect ratio problems. It will show the current maximum aspect ratio, and the target can be entered, as shown in Figure 163. Once finished, the new current maximum aspect ratio is shown. Be sure to inspect the model to make sure it fixed all the problematic areas and did not make some areas of the mesh incorrect. This is the same feature as the fix aspect ratio on the mesh repair wizard. This tool has largely been replaced by the wizard.

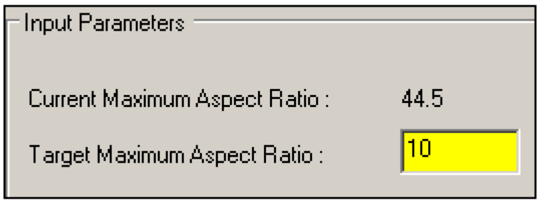


Figure 163: Fix aspect ratio

Global merge

The global merge command is used to merge nodes that are closer together than a specified tolerance. The tool is shown in Figure 164. The merge tolerance can be set. **Preserve Fusion** is on by default. This is to prevent nodes that are part of a small thickness (less than the tolerance) from being merged. However, if you have this option ticked, it may prevent merging that you actually want to occur. After running the Global Merge tool, if nodes were not merged that you would like to be merged, turn off the Preserve Fusion and run it again. Once complete, make sure the model is satisfactory and it did not collapse areas that were unintended.

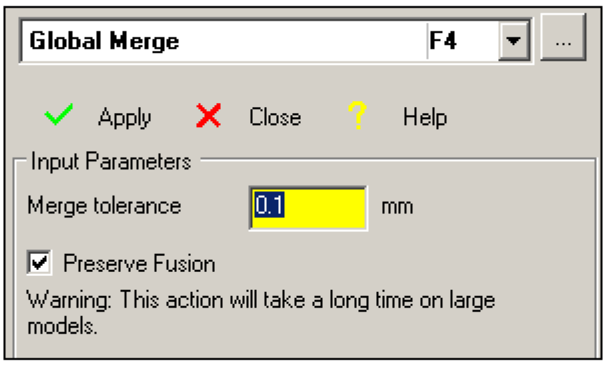


Figure 164: Global merge

Merge

This command merges nodes manually. This is a very commonly used tool and is shown in Figure 165. The first node you picked is the node that is retained, so remember to pick the node(s) you wish to remove second. All the nodes in the second box **Nodes to merge from** are merged into the first. Normally this is one node, but it could be several. All of the selection methods are available, including holding the CTRL key and band selecting.

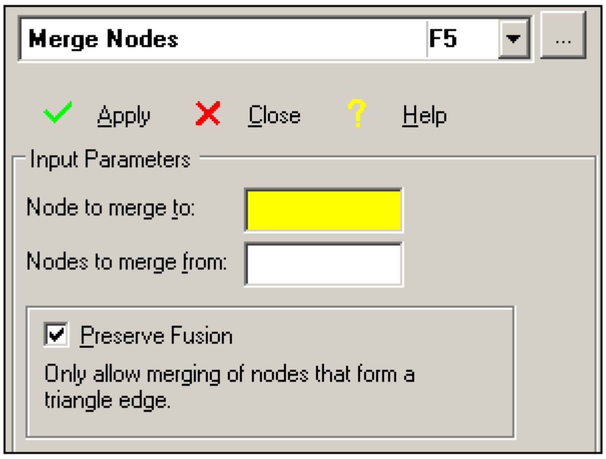


Figure 165: Merge nodes

When using the band selection method, also use the "Select facing items only" filter. In this way, you will not pick nodes hidden behind the elements that you can see.

Programming the mouse with the "Mouse Apply" action or using the context menu (right-click) is a useful way to "**Apply**" the merge. It is the first item in the context menu.

In Figure 166, to fix the high aspect ratio element, click on the node marked 1 first, then on the node marked 2. In this case the node marked 1 had to be selected first as it formed a corner. In many cases, it will not matter which one is selected first.

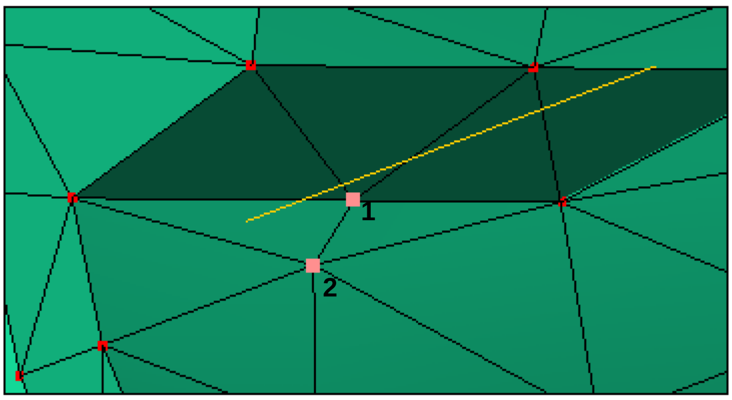


Figure 166: Merging nodes

Swap edge

The swap edge tool, shown in Figure 167, allows you to swap a shared edge between two elements, usually to improve the aspect ratio of those elements. In any mesh, two adjacent, triangular elements (when combined) form a quadrilateral, divided along one shared edge. The end result of the Swap Edge tool is a division of the quadrilateral along the alternative available edge. It does not matter in which order you select the two elements. The elements are then recreated using the same four nodes of the original selection. See Figure 168. Depending on the geometry, the box **Allow remesh of feature edges** must be checked in order to create an element. If the Apply button is clicked and an element is not created, check this box.

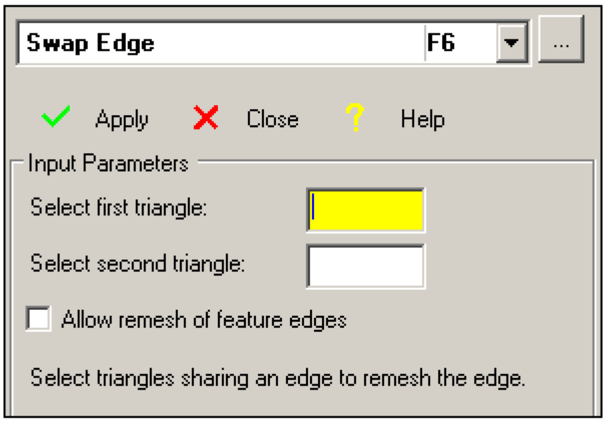


Figure 167: Swap edge

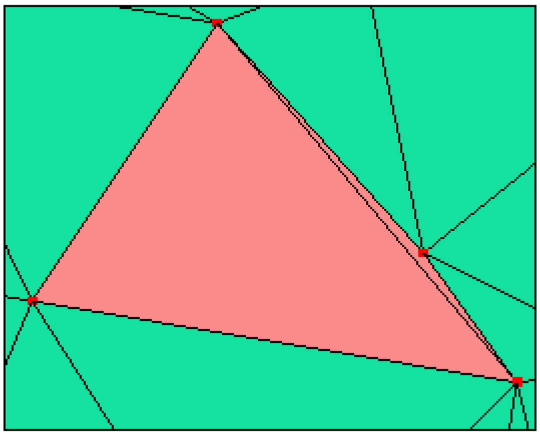


Figure 168: Swap edges to fix high aspect ratio element problems

Match node

The match node tool, shown in Figure 169 attempts to match nodes from one side of the wall thickness on a Fusion model to elements on another side of the thickness. The purpose of this tool is to improve the mesh matching. Use the check box to put new nodes on a specified layer. Enter in the name of the layer.

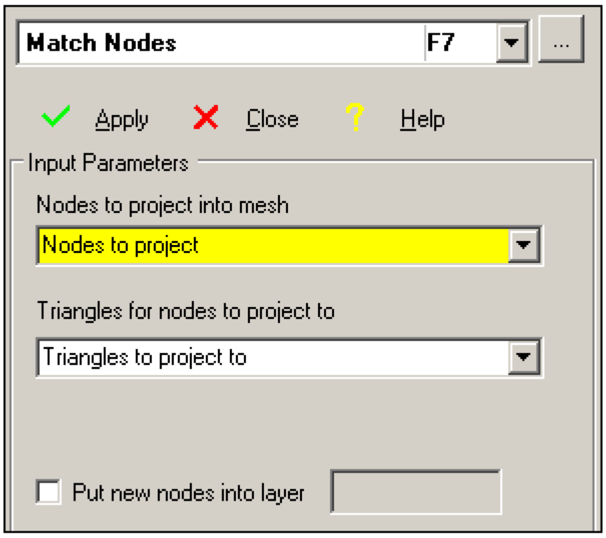


Figure 169: Match node

Remesh area

The remesh tool allows you to select an area of the part and change the mesh size in that area. For Fusion meshes, elements on both sides of the cross section should be selected. Then the tool will re-match the elements with the new mesh density.

When a group of elements is selected, the **Target edge length** box is updated with a value based on the selection. This can be a guide for entering a new number. See Figure 170. You should avoid making too large a change because the transitions between mesh sizes will not be good. The transition can be helped to some degree with the **Smooth Nodes** tool, which will be explained later. Figure 171 shows a sample of an original mesh, the selected mesh, and the remeshed elements.

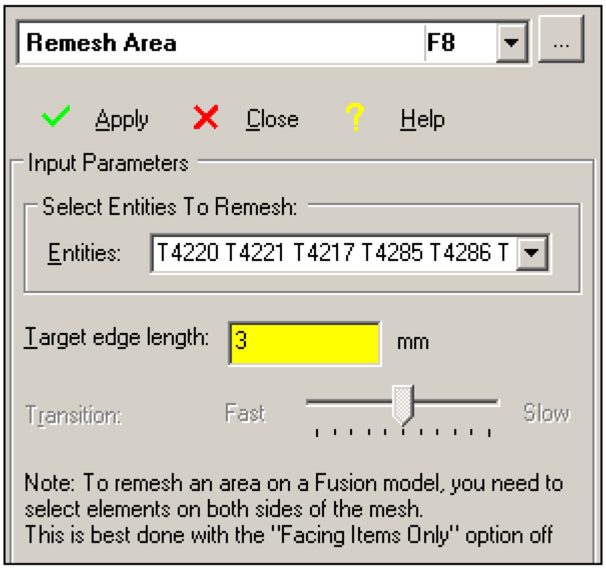


Figure 170: Remesh for a Fusion model

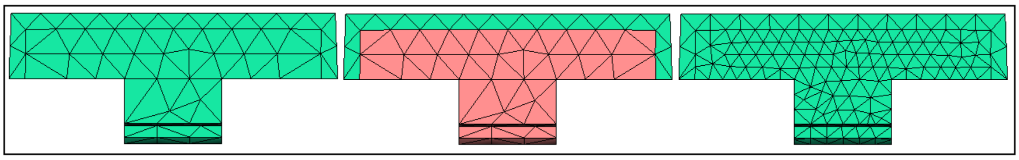


Figure 171: Remeshing an area of a Fusion part

Insert nodes

Inserting a node is a common method for fixing local mesh problems. See Figure 172. It is often done in the process of fixing aspect ratio problems or just to improve the mesh in a local area. There are three ways in which a node can be created including:

- Midpoint of a triangle - This is by far the most common method and is the default.
- Center of a triangle.
- Center of tetrahedron. - This is only available when editing a 3D mesh.

A node is inserted based on your selection, when you click on two nodes on the same element(s). Any additional elements that need to be created to keep connectivity will be created and properties assigned. Figure 173 shows how the Insert Nodes tool creates the node and connects the mesh by creating the additional elements necessary.

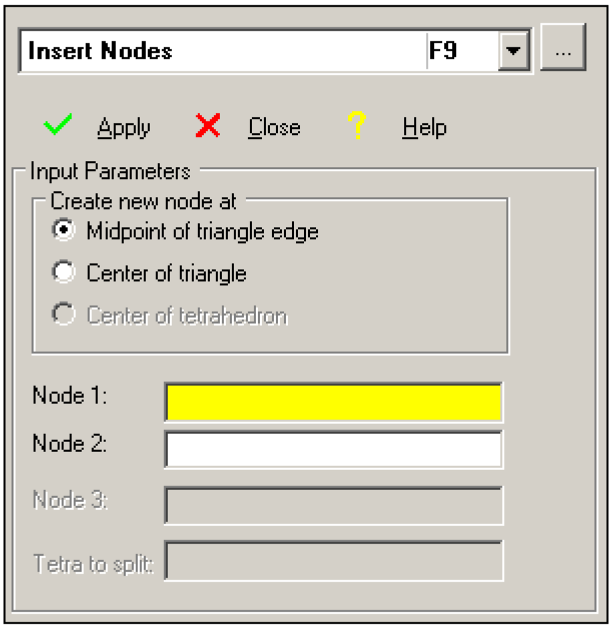
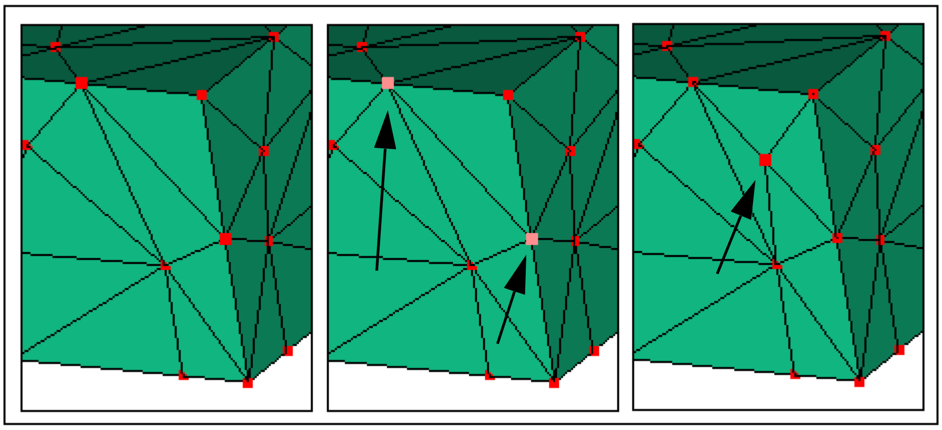


Figure 172: Insert nodes



Original mesh **Nodes selected** **Node inserted**

Figure 173: Inserting a node

Move nodes

The move node tool is shown in Figure 174. Moving a node can be done in three ways:

1. Click on a node and drag it, as shown in Figure 175.
   - Drag the node to the boundaries of the elements it forms.
   - If you move the node to the location of another node, the nodes will be merged.
   - Once the new location is found by dragging, click **Apply** to finish the move.
   - If the node creates a feature edge, you can only drag the node along the edge.
   - If the node creates a feature corner, you cannot drag the node.
2. Enter absolute coordinates.
3. Enter relative coordinates.
   - Relative coordinates are based on the existing location of the node.

If you are moving a node by coordinates, it is best to select the node by banding. Make your banding selection using the Facing Items Only filter (**Edit ➡ Banding Selection ➡ Facing Items Only**).

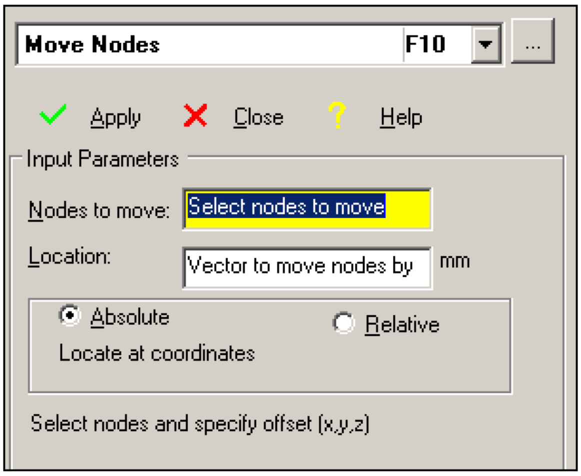


Figure 174: Move node

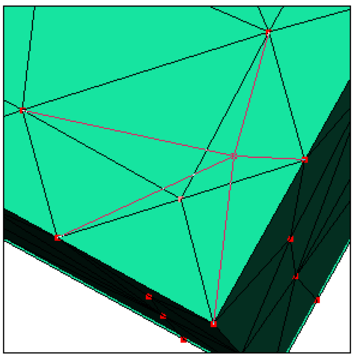


Figure 175: Dragging a node to move it

Align nodes

Align Nodes is a useful tool to straighten up a mesh, see Figure 176. Sometimes in the process of meshing a geometry file, an edge that should have been straight is not. There are many possible reasons why nodes may need to be aligned.

To align nodes, two reference nodes need to be picked first. These nodes form the line definition. All nodes picked after the first two nodes align with the first two. In Figure 177, a node is aligned in two directions to form a better corner.

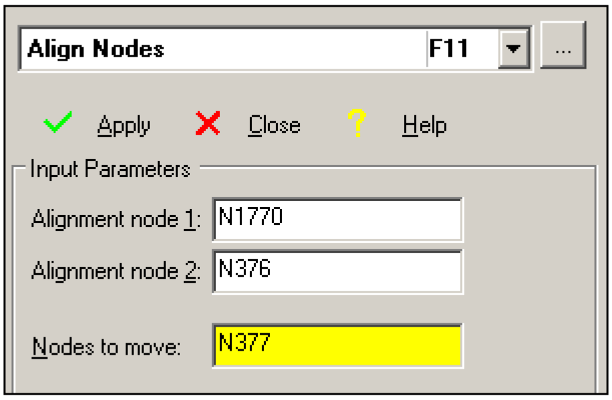


Figure 176: Align nodes

It may be very useful to pick a whole series of nodes at one time to move to a line. You can do this with banded selection. Depending on the situation, the part should be rotated so you are looking at the nodes and attached elements that are to be aligned. Other times it is better to rotate the part so you are looking down the line of nodes. Banding filters of **Facing items only**, or **Enclosed items only** may need to be turned on or off depending on the situation.

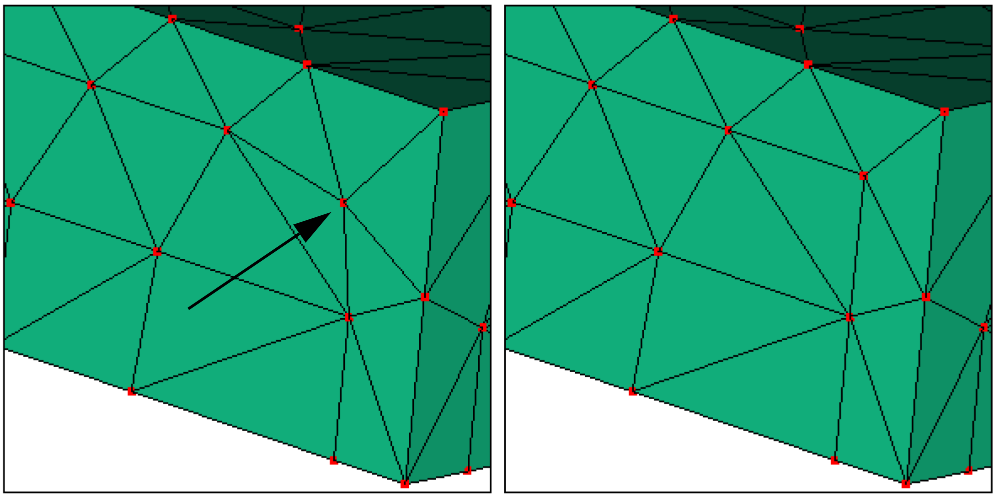


Figure 177: Nodes being moved to a line

Orient elements

Elements in models usually need to have their orientation corrected for a couple of reasons:

- The original mesh is not correct to begin with; or
- In the process of fixing the mesh elements became un-oriented.

Using the mesh diagnostics tools, you can find the location of the orientation problem. For Fusion models, usually the quickest way to fix the orientation is to use the command **Mesh ➡ Orient All.** However, sometimes this does not fix the problem, so you can use the **Orient Elements** tool, shown in Figure 178. There are several ways this can be used.

Method 1

1. Click in the **Reference** box, then click on an element in the part that has the correct orientation.
2. Click **All connected elements to reference element**.
3. Click the **Search** button.
    - ➤ All connected elements should pick the entire model.
4. Click on **Align orientation**.
5. Click on **Apply**.

Method 2

1. Click the Reference box.
2. Click on an element in the part that has the correct orientation in a region you would like to orient, such as element T7377 in Figure 179.
    - ➤ Elements in that region will pick elements on the same plane, as the elements picked have the same current orientation. This technique is best used when an entire region has the incorrect orientation.
3. Click **Search**.
4. Click on **Flip orientation.**
5. Click on **Apply**.

Method 3

1. Select all the elements that need to be flipped. Use any combination of selection tools necessary.
    - ➤ Click on individual elements.
    - ➤ Band select with facing elements only filter on.
2. Click on **Flip orientation**.
3. Click **Apply**.

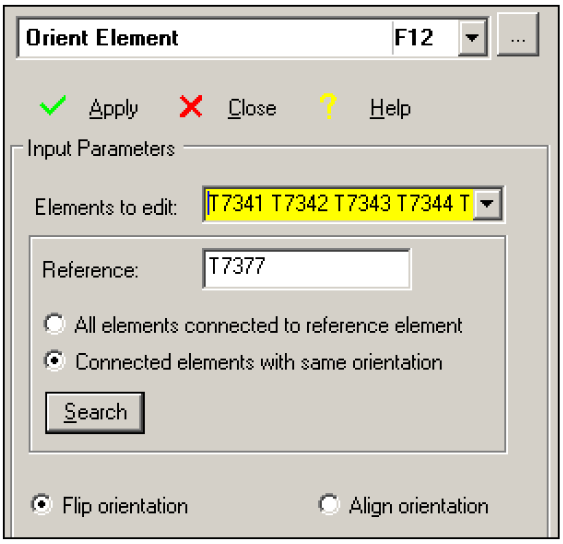


Figure 178: Orient elements

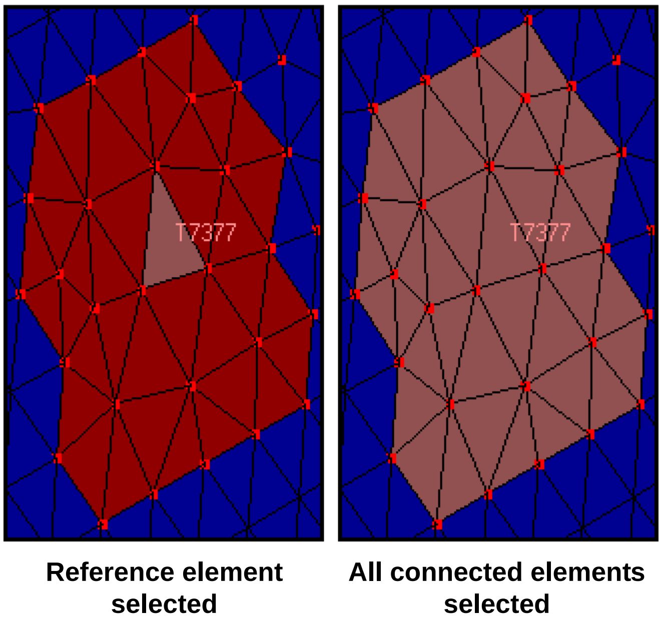


Figure 179: Selecting elements to be oriented by reference element

Fill hole

The Fill Hole tool, shown in Figure 180, is useful on Fusion models where there is a hole in the mesh. Holes are easily found with the **Free edges** diagnostic.

Use the Fill Hole tool as follows:

1. Zoom in on a free edge.

2. Click on one node that makes the free edge.

3. Click the search button to find all the other nodes attached to the free edge.

4. Click **Apply**. Figure 181 shows a hole before and after being filled.

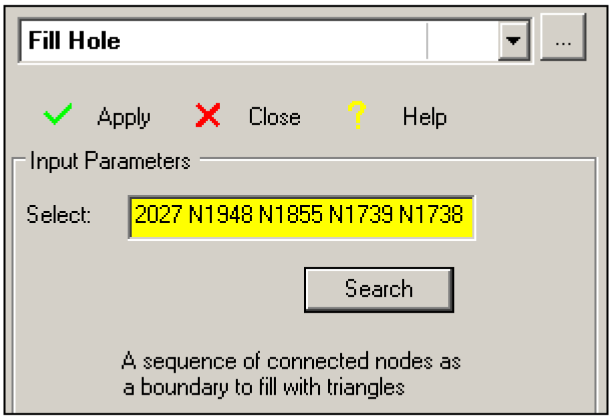
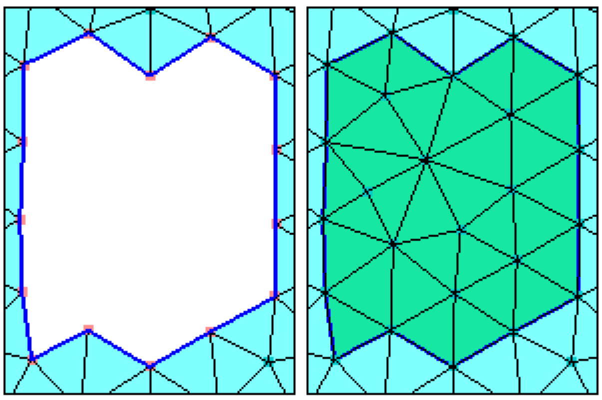


Figure 180: Fill hole

If the hole is large and the area you want to fill is not planar, you may need to fill the hole in stages. You can select or deselect nodes by any of the normal means to define the hole you want to fill.



Selected free edge **Filled hole**

Figure 181: Filling a hole

Stitch free edges

An edge can be stitched when the distance between two free edges is less than the specified tolerance. The default tolerance is 0.1 mm. The selection of nodes to be stitched can be all of the nodes on the model, as this command will only use nodes that are on a free edge and is less than the tolerance specified. Figure 182 shows the stitch free edges tool with a list of all the nodes in the model with a specified tolerance. Figure 183 shows a free edge before stitching, all the nodes in the models selected and the nodes stitched to remove the free edge.

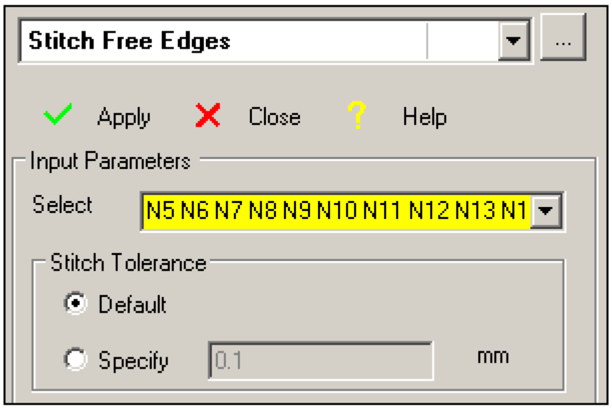
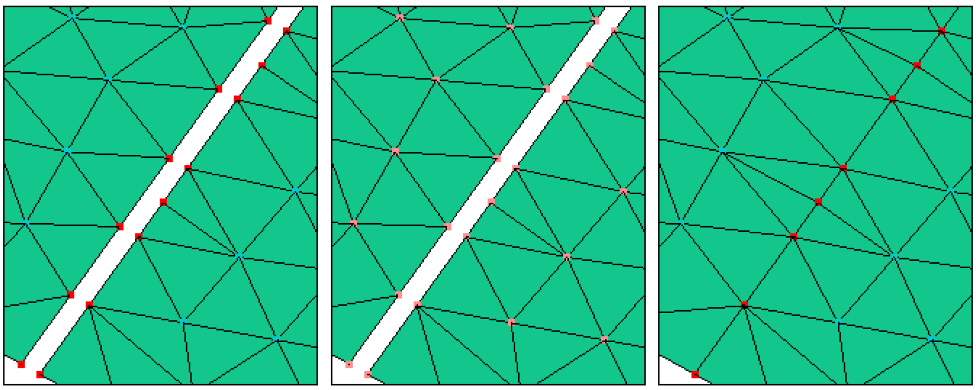


Figure 182: Stitch free edges



Red nodes on free edge to be stitched | **All nodes selected in model** | **Nodes on free edge less than the tolerance stitched**

Figure 183: Stitching before, during and after

Create regions

The Create Regions tool will convert the entire part mesh model, and create regions, based on the mesh or an STL file. There are two tolerances that can be used:

- Planar tolerance.

  The elements within the planar tolerance are used to create a region.

- Angular tolerance.

  This works better on parts that are mostly curved. With this tolerance, elements with the defined angle are used to create a region. You may want to run this command several times adjusting the tolerance to find one that works best.

This command would normally be used to create geometry so it can be manipulated in Synergy. Normally, this will be a matter of setting local mesh densities to provide the most control over the mesh density of the part. This tool is rarely needed today because of all the ways meshes can be controlled.

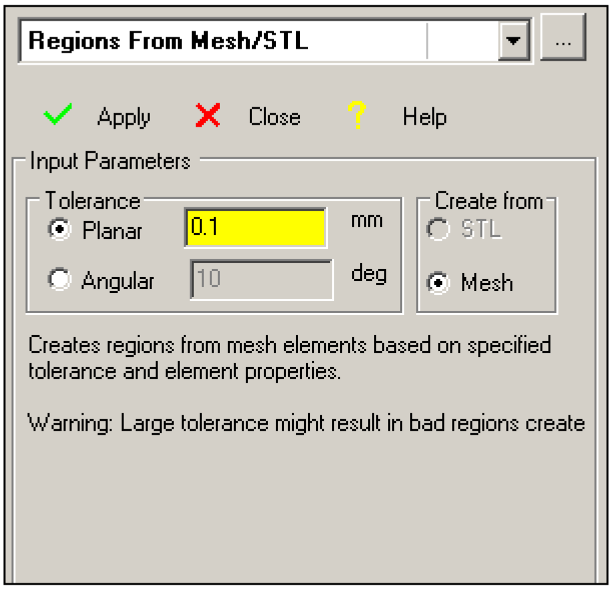


Figure 184: Create regions

Smooth nodes

The Smooth Nodes tool, shown in Figure 185, averages the spacing between the nodes to make the mesh more uniform. This can be useful to help smooth the transition between a re-meshed area and one that was not re-meshed. To use this tool, select the nodes you want to smooth. Clicking the **Preserve feature edges** will prevent corners of the part from being distorted. An example of smoothing is shown in Figure 186.

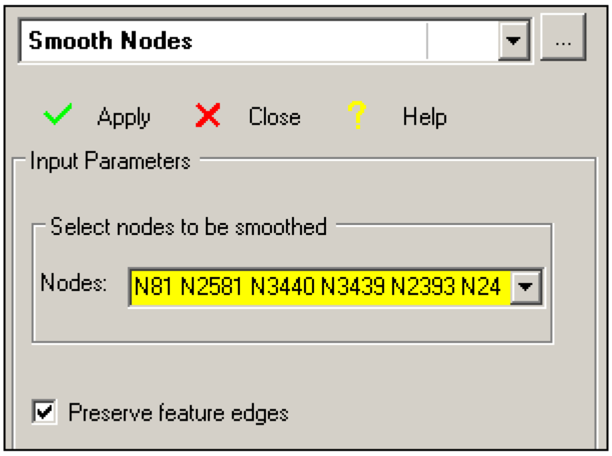


Figure 185: Smooth nodes

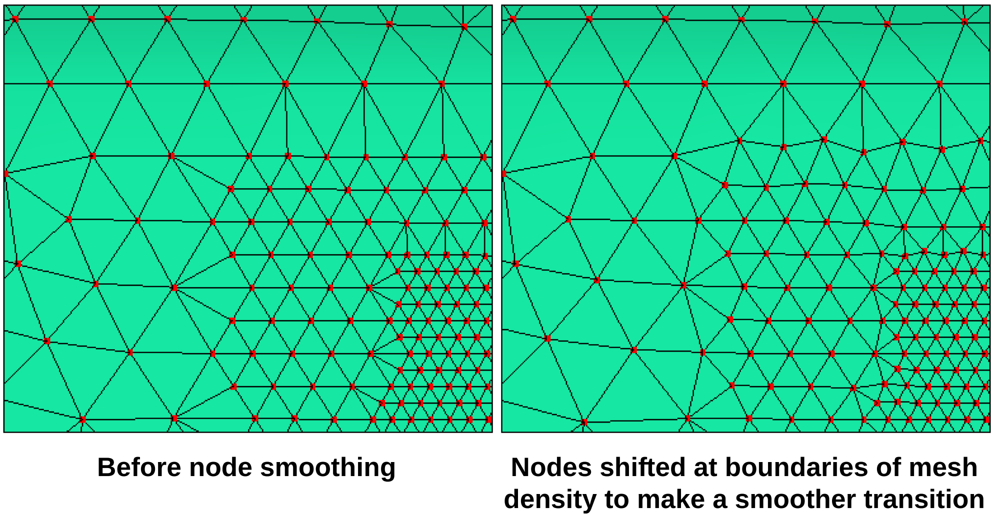


Figure 186: Smoothing a mesh

Create beams

Beam elements for feed systems, water lines, etc. can be created directly as beam elements. Two end nodes need to be defined first.

Creating Beams

1. Enter the coordinates that define the beginning and end of the beam.
   - Normally a node is clicked to define the coordinate.
2. Enter the number of beam elements to create.
3. Click the icon ... to select the beam type with the properties defined properly.

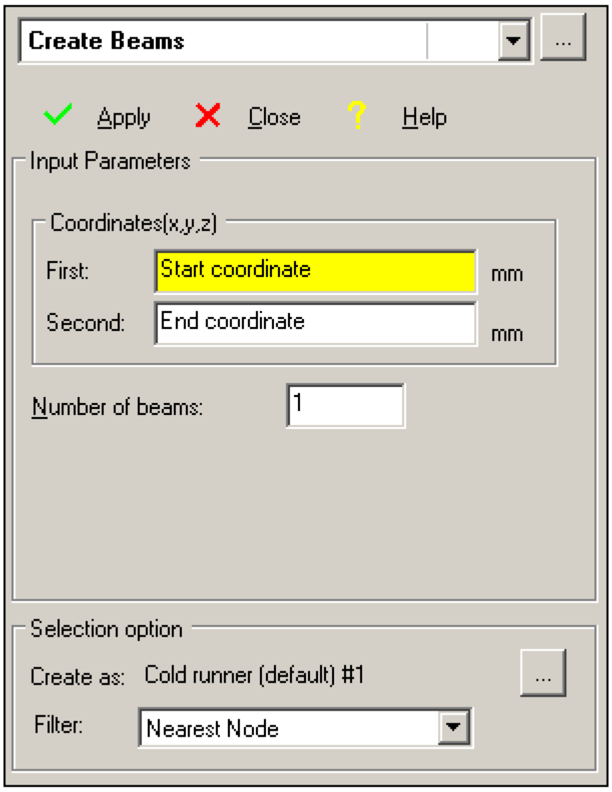


Figure 187: Creating a beam element

The number of elements that you need to create depends on what kind of beam you are creating and how long it is. Gates should have a minimum of 3 elements. Any runner section should have at least 3 also. The length-to-diameter ratio for beams should be 2.5:1 as a minimum.

Create triangles

Generally, you should not need to manually create triangles. Most mesh clean-up tools will automatically create the needed triangles. To create a triangle, click on three nodes, as shown in Figure 188. Leaving the **Inherit properties from neighbors** box checked (its default position) will automatically assign properties of the new element to its neighbors. Otherwise you can manually assign the properties.

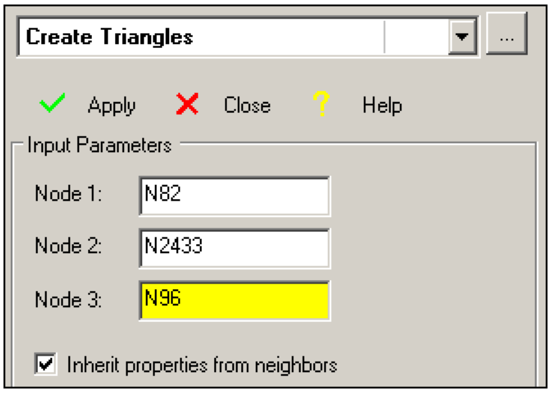


Figure 188: Create Triangles

Delete entities

For general model clean-up, Delete Entities is also not a recommended tool. If it is necessary, use any of the selection techniques to pick any entity to delete, as shown in Figure 189. In addition to using this tool via the Toolbox, the Delete key on your keyboard also deletes entities once you have selected them, however none of the Toolbox tools can be open.

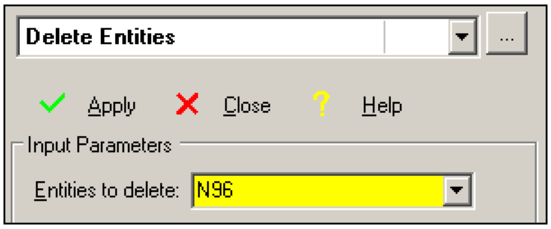


Figure 189: Delete entities

Purge nodes

In the process of fixing models, sometimes elements are deleted but the nodes are not. Deleting nodes is done in one step with the Purge Nodes command, as shown in Figure 190. It deletes any node that is not connected to an element.

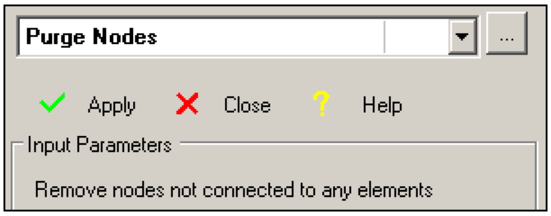


Figure 190: Purge nodes

Create tetras

Generally, you should not need to manually create tetrahedral elements. Most mesh clean-up tools will automatically create the needed tetras. To create a tetra, click on four nodes, as shown in Figure 191. Leaving the Inherit properties from neighbors box checked (its default position) will automatically assign properties of the new element to its neighbors. Otherwise you can manually assign the properties.

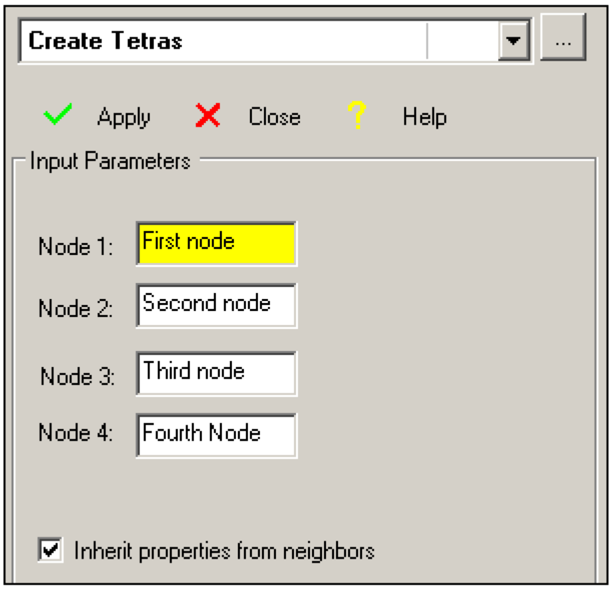


Figure 191: Create tetrahedral elements

Remesh tetras

Remeshing a tetrahedral mesh is done normally to increase the number of layers through the thickness in a local area. To remesh an area, select the elements to be remeshed and enter the number of layers, as shown in Figure 192. Generally, if the number of layers gets increased, the Surface mesh must be made smaller. Normally it's best to reduce the surface mesh edge length.

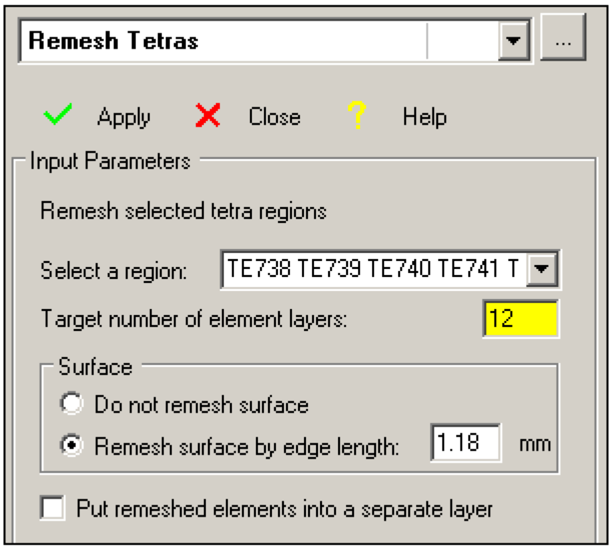


Figure 192: Remesh tetras

Verify mesh is clean

Once manual repairs are done, the mesh should be clean and ready for analysis. Use mesh statistics to verify the mesh is clean. There may be a point where cleaning the mesh any more is a waste of time. The mesh problems left to fix will take a long time in relation to the potential problems in the analysis.

Decide when to stop fixing

ALL intersections, overlaps, free and non-manifold edges, zero area elements, plus orientation **must** be fixed. The aspect ratio should be below 6:1, but if there are a few scattered elements in non-critical areas, slightly higher values are acceptable for Flow. If a cool and warp analysis is required, the mesh must be very good.

Clean up the layers

Normally during the process of mesh cleanup extra layers get created and used. After cleanup remove any extra layers and put the elements and nodes back in their original layers. One possible way is to:

- Make the Triangles layer active.
- Highlight all the layers to be deleted one at a time, and click the Delete button. Say **Yes** to move the entities to the active layer.

There will probably be nodes on the Triangles layer. To remove the nodes use the **Select By** command. Pick nodes to select, highlight the nodes layer and click Assign.

How to fix common problems manually

When automatic fixing methods do not fix all the problems and manual methods are needed, use the mesh diagnostics to display and find the problem areas. Table 25 summarizes the tools necessary to fix many common mesh problems.

Table 25: Methods for fixing mesh problems

Problem	Possible solutions
Low mesh match ratio	• Decrease global or local edge length and mesh again.
Thickness	**1.** Increase the mesh density. **2.** If the mesh density is generally good, re-assigning the thickness, after other problems fixed. **Note:** do not change the thickness much. A large change will cause problems for Cooling and Warp.
Connectivity	**1.** Stitch free edge. **2.** Global merge. **3.** Merge node.
Intersections and overlaps	**1.** Try **Auto Repair** first. Check to make sure the auto repair did not create any problems. **2.** Merge nodes. **3.** Delete elements. then fill hole.
Free or Non-manifold edge	**1.** Create nodes. **2.** Merge nodes. **3.** Delete entities (elements). **4.** Fill hole.
High aspect ratio	**1.** Merge nodes. **2.** Swap edge. **3.** Insert node. **4.** Move node. **5.** Align nodes.
Un-oriented elements	**1.** Orient all. **2.** Orient element.

Generally meshing the part with a smaller global or local edge length will go a long way in fixing many mesh problems. It then becomes a compromise between computer time for each analysis and initial preparation time.

What You've Learned

Preparing a good finite element model is one of the most important steps in the analysis process. The primary steps for preparing a finite element model include:

1. Create the CAD translation model.
    - A geometry format such as IGES is preferred.
    - With Moldflow Design link, many formats can be imported including native CAD formats.
2. Import the CAD model.
3. Set the mesh density.
4. Generate the mesh.
5. Evaluate the mesh.
6. Determine if the initial mesh is good, if not remesh or re-create the CAD model.
7. Clean up the mesh.

    7.1. Use the Mesh Repair wizard.

    7.2. Validate the repair of the wizard.

    7.3. Use mesh diagnostics to find problem areas not corrected by the wizard.

    7.4. Manually fix the mesh.

    7.5. Verify the mesh is clean.
8. Create the 3D or Midplane mesh if necessary.

    8.1. Clean the 3D or Midplane mesh.

CHAPTER 10

Modeling Tools

Aim

To learn about the modeling tools available inside Synergy. Learn how to use them to create geometry and/or features in the model to be studied such as part features or gates.

Why do it

Even though MPI is not a CAD program (it was not developed to design parts), it is sometimes necessary and convenient to create or add geometry to the model inside MPI for running an analysis. Changing the geometry inside MPI is a quick and convenient way to have the changes made in the model.

Overview

In this chapter, you will be learn about mesh elements and practice modeling within Synergy. The concepts that you will review are:

- Terminology.
- Properties.
- Features likely to be modeled within Synergy.

After you determine that a feature needs to be added to the model you will need to do the following:

1. Decide if the feature should be designed as a midplane or fusion mesh.
2. Create lines (or nodes) that will be used to define the features.
3. Set the property type depending on the model type, midplane or Fusion.
4. Create the regions.
5. Mesh the regions.
6. Check the mesh quality, and if needed do the adjustments.

The geometry creation steps will be discussed in detail in this unit.

Theory and Concepts - Modeling Tools

Terminology

Mesh

Injection molding simulation involves solving the governing equations of mass, momentum, and energy numerically over the physical domain. The numerical implementation involves dividing the physical domain into a number of sub-domains, or elements. The dependent variables: velocity, pressure, and temperature are approximated within each element. In short, the continuous domain is broken into many connected sub-domains and the dependent variables are approximated over the whole domain. Mesh density is the number of elements per unit area. In general, the more elements in the mesh, the more accurate the analysis results, at the expense of longer calculation times.

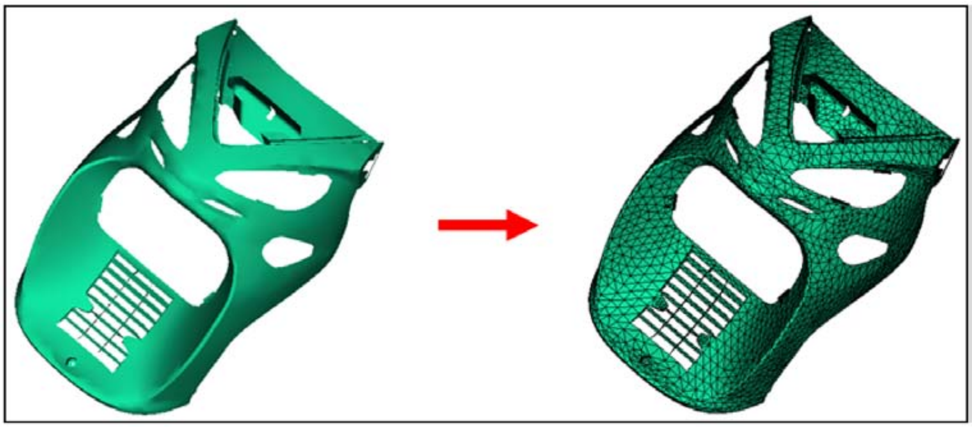


Figure 209: Mesh

MPI supports 3 types of mesh for part representation: midplane, surface (also known as Fusion, or dual domain) and 3D. While runners and cooling channels are represented using beam elements; these are also known as 1D element.

Element

An element is a single sub-domain of a finite element mesh. Elements typically used in molding simulation are:

- Two-noded linear elements (beams), used to represent runners, cooling channels, and connectors.
- Three-noded triangular elements (shell), used to represent surfaces for midplane and Fusion.
- Four-noded tetrahedral elements, used to represent solids for 3D analysis.

The properties of the element are assumed to be homogenous.

Node

In the context of model geometry, a node is a coordinate position in space. Nodes are saved with a model. When modeling, you typically create nodes and then use nodes to create curves and regions. Nodes take on special significance when you assign attributes to them. For example, you might assign an injection location, a coolant inlet, a constraint, or a load to a model node.

In the context of mesh, when you mesh a model, model nodes are converted to mesh nodes and additional mesh nodes are generated. Mesh nodes are the vertices of midplane, surface, 3D mesh elements, and the ends of beam elements. The graphic below shows mesh nodes at the vertices of a triangular element and a beam element.

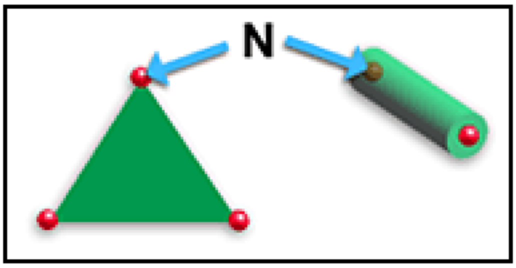


Figure 210: Nodes

✎ When you run an analysis, certain results are recorded at mesh nodes.

Curve

A curve is a line in three-dimensional space. A curve can be straight or contain bends. This includes:

- Line: a straight curve defined by two end points.
- Arc: a section of a circular curve defined by one of two methods:
- A center point and arc and angle.
- Three points on a plane.
- Spline: a cubic spline interpolation on a supplied set of points.

Region (flat surface)

A region is a planar space defined by a consistent set of curves. The curves connect but must not intersect as shown in Figure 211.

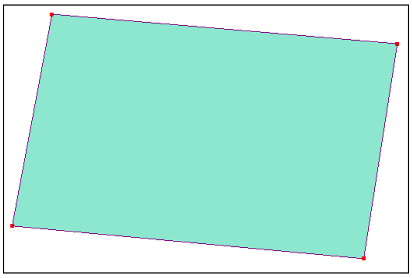


Figure 211: Region

Surface

A rectangular surface defined by a grid of points which represent the control points of a NURBS definition of a surface. NURBS is a Non-Uniform Rational B Spline, a common mathematical representation in the CAD industry. NURBS surfaces are not often used, because they are limited to a rectangular shape. Trimmed surfaces are more useful and common. A trimmed surface is a NURBS surface with a set of defined curves in the surface. This provides an easy way to define non-rectangular representations (patches) of surfaces. Surfaces provide important information for meshing operations, which allow much better meshes to be achieved than via facet-based mesh input.

Assigning Properties

Depending on the type of element selected the program will customize to display the properties only available for such kind of element. The properties are divided for:

- Beams or curves
- Triangles
- Tetras

The available properties in the list will depend on the type of molding process set in the interface: thermoplastics injection molding (default), Reactive molding, Injection-compression, Gas- Assisted injection molding, etc.

Beam elements or curves

When assigning a property to a curve or beam element, the program will display the following options:

- Baffle.
- Channel.
- Cold runner.
- Connector.
- Hose.
- Hot runner.
- Part beam.
- Bubbler.
- Cold gate.
- Cold Sprue.
- Critical dimension.
- Hot gate.
- Hot sprue.
- Thermal pin.

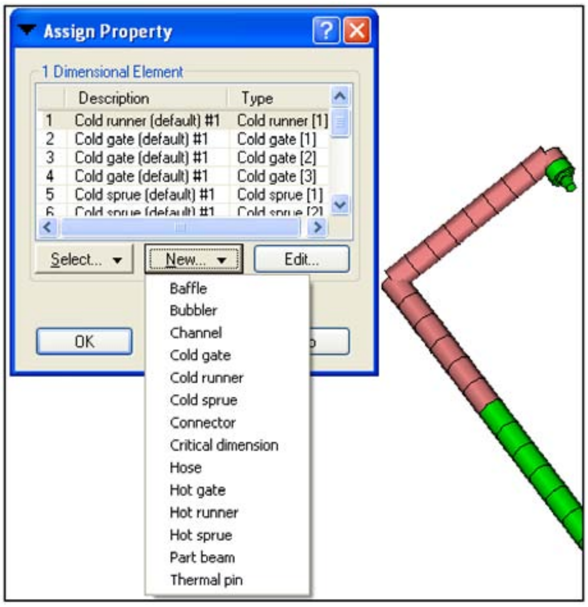


Figure 212: Properties list for beam elements and curves

Triangle elements

When assigning a property to a triangle element, the program will display the following options:

- Cold gate surface (Fusion).
- Cold gate surface (Midplane).
- In-mold label.
- Mold block surface.
- Mold insert surface.
- Part insert surface (Fusion).
- Part insert surface (Midplane).
- Part surface (Fusion).
- Part surface (Midplane).
- Parting surface.

The properties with **(Midplane)** in their name are available in both Fusion and Midplane models. Properties with **(Fusion)** in their name are available in Fusion models only.

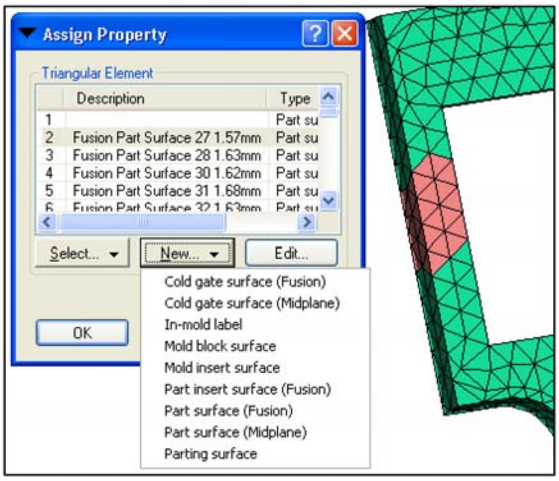


Figure 213: Properties list for triangle elements of a Fusion model

Tetrahedral elements

When assigning a property to a tetrahedral element, the program will display the following options:

- Core (3D).
- Hot runner (3D).
- Part (3D).
- Part insert (3D).

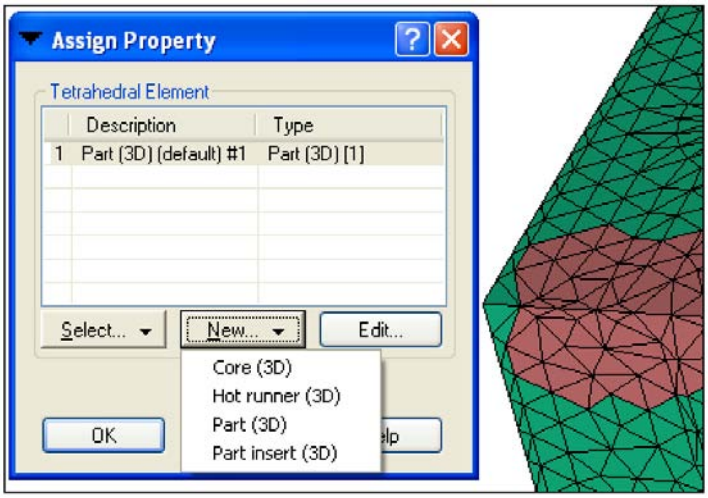


Figure 214: Properties list for tetrahedral elements

Features likely to be modeled within Synergy

Users may use the modeling tools to create simple geometries inside Synergy such as:

- Runners.

- Gates, including fan gates.
- Sprues.
- Cooling lines, (circuits).

The geometries are based on nodes, curves and or regions. In general, the geometries are features that are straightforward to create inside the interface. For complex geometries we recommend the user to model such sections in the CAD program since their modeling functionality is more advanced.

Use of modeling tools

Modeling Menu

The tools to create geometries inside MPI can be accessed from the modeling menu. The modeling menu options are shown below:

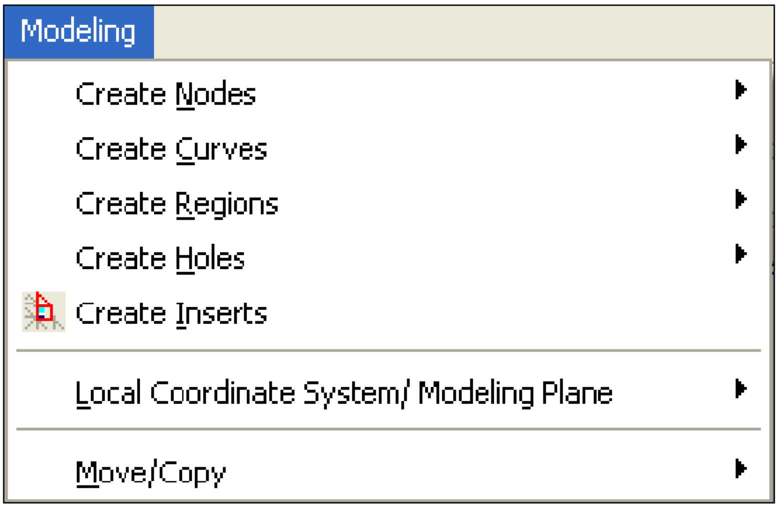


Figure 215: Modeling Menu

Create nodes

A model node is a special coordinate position in space. Nodes, along with curves and regions, are the building blocks of a model. Nodes acquire special significance when you assign boundary conditions to them, for example, injection locations. The dialog to create nodes is shown in Figure 216.

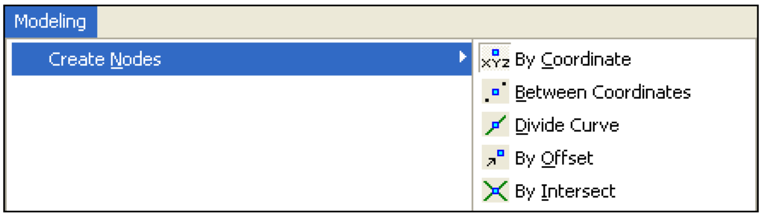


Figure 216: Create nodes menu

To create a node

1. Select **Modeling ➧ Create Nodes...** to open the Create Nodes tool, or click the Create nodes icon  from the toolbox menu, shown in Figure 217.

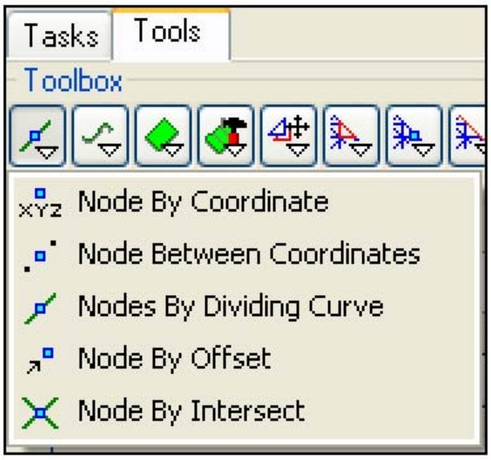


Figure 217: Toolbox - Create node

2. Select the option from the pull down menu, shown in Figure 217 that corresponds to the node creation method you plan to use.
3. Enter the information as required.
4. Click **Apply** to create nodes.
5. Click **Close** when you finish creating nodes.

Node creation options

You can create nodes using the following options:

Table 32: Node creation options

Option	Description
Coordinate	Allows you to specify the coordinate of the new node.
Between Coordinates	Allows you to create the number of nodes that you would like between two specific locations that you Specified.
Divide Curve	Allows you to create nodes on a curve which will be equally separated.
Offset	Allows you to create a node vector away from the reference point. You may create more than one node at once.
Intersect (curves)	Allows you to create a node at the intersection of two curves.

Create curves

A curve is a geometric line on your model. Curves can be straight lines between two points, or arcs made of three or more points. Curves, nodes, and regions, are the building blocks of a model. The dialog to create curves is shown in Figure 218.

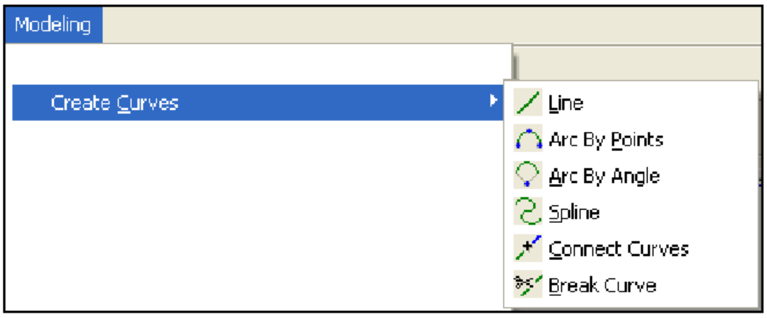


Figure 218: Create curves menu

To create curves:

1. Select **Modeling** ➡ **Create Curves...** to open the Create Curves tool.
2. Or click  from the toolbox menu.

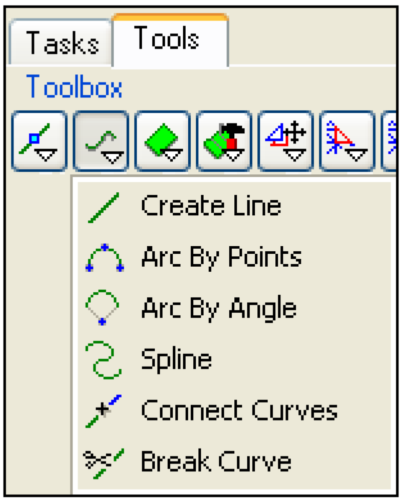


Figure 219: Toolbox - Create curve

3. Select the option from the pull down menu, shown in Figure 219 that corresponds to the curve creation method you plan to use.
4. Enter the information as required.
5. Click **Apply** to create curve.

6. Click **Close** when you finish creating curves.

New curves will be created in the active coordinate system. The values shown in dialog input boxes always reflect the active coordinate system. This could be a global or local coordinate system.

You can create curves using the following options:

Table 33: Curve creation options

Curve options	Description
Line.	Allows you to specify the origin and the end of the line to be created.
Arc by Points.	Allows you to select three points which the program will use as best fit for the arc or circle to be created.
Arc by Angle.	Allows you to create a circle or an arc by specifying its center, radius, starting and ending angle.
Spline.	Allows you to create a line that will pass through all the points providing the best fit softening the edges.
Connect Curves.	Allows you to connect two curves with the fillet.
Break Curve.	Use this command to break curves into segments. The program uses the intersection of curves to be the reference point to break them.

Create regions

The menu to create regions is shown in Figure 220.

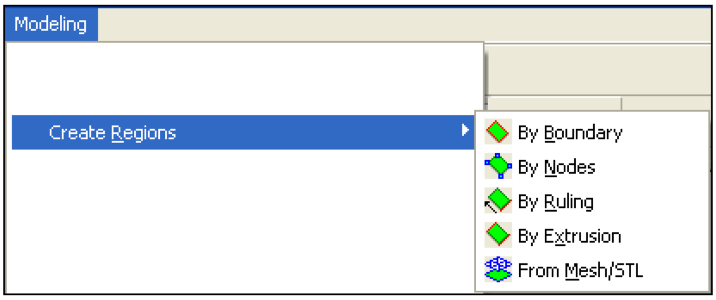


Figure 220: Create regions menu

To create a region:

1. Select **Modeling** ➡ **Create Regions…** to open the Create Regions pane.

2. Or click  from the toolbox menu.

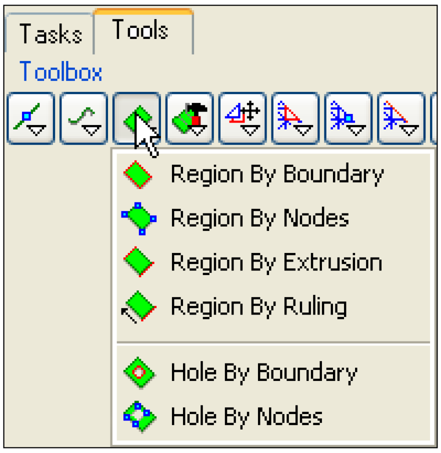


Figure 221: Toolbox - Create region

3. Select the option from the pull down menu, shown in Figure 221, that corresponds to the region creation method you plan to use.
4. Enter the information as required.
5. Click **Apply** to create region.
6. Click **Close** when you finish creating region.

You can create regions using the following options:

Table 34: Region creation options

Region options	Description
Region by Boundary.	Allows you to select the curves that delimit the region to be created. After selecting one curve you may click on the search icon to highlight the rest of them.
Region by Nodes.	Allows you to create a region by selecting the nodes that will delimit the out boundary.
Region by Extrusion.	Allows you to create a region by extruding a curve in a specific direction.
Region by Ruling.	Allows you to create a region between two curves.

Hold the **Ctrl** key while you select the nodes to add them to the current selection.

Create holes

The dialog to create regions is shown in Figure 222.

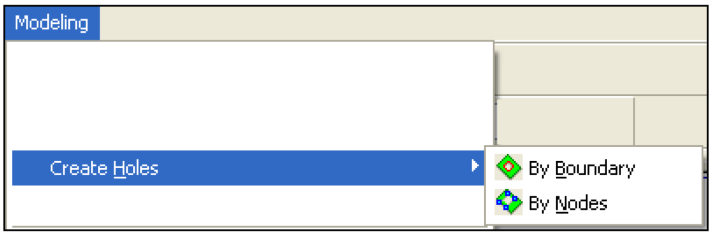


Figure 222: Create hole menu

To create a hole:

1. Select **Modeling** ➡ **Create Holes...** to open the Create Holes pane.
2. Or click  from the toolbox menu.

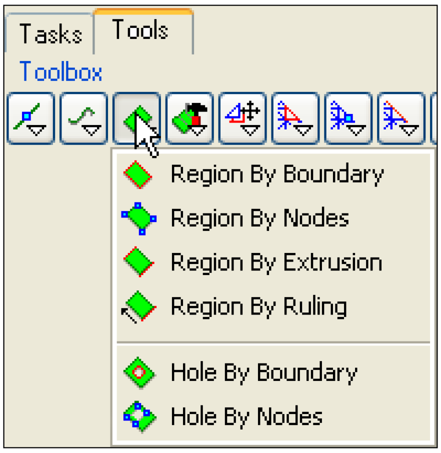


Figure 223: Toolbox - Create hole

3. Select the option from the pull down menu, shown in Figure 223, that corresponds to the hole creation method you plan to use.
4. Enter the information as required.
5. Click **Apply** to create the hole.
6. Click **Close** when you finish creating the hole.

You can create holes using the following options:

Table 35: Hole creation options

Hole options	Description
Hole by Boundary	Allows you to create a hole in a region by selecting a curve to delimit the hole.
Hole by Nodes	Allows you to create a hole in a region by selecting the nodes that highlights the hole.

Specifying coordinates

To select or specify coordinates and other model entities

This topic provides you with two alternate methods for selecting or specifying coordinates and other model entities in MPI, and describes the difference between absolute and relative coordinates.

Table 36: Methods for Specifying coordinates

Method 1	
Click on the model	When you are working in a modeling dialog such as: Create Nodes, Create Curves, and Create Regions, you are prompted to select or specify coordinates, curves, regions, or other model entities. Often the easiest selection method is to move your cursor out of the dialog and over to the model, where you can click directly on the desired coordinate or entity. You may rubber band-select model entities as well.
Method 2	
Type into the dialog input field	In cases where the model-click method is not easy or not available, you should type values into the dialog input boxes. • Coordinate positions. ➤ To specify a XYZ coordinate position, use this format: 30 -60 90 (A space or comma can be used as a delimiter) • Curves and regions. ➤ To specify curves and region, type the appropriate label, for example, C1, C2, R1, R2.

If the z coordinate is zero, type just the x, and y values (e.g.: 30 -60). If the Y, and Z coordinates are zero, type just the x value (e.g.: 30).

Using absolute or relative coordinates

Inside MPI you can specify coordinates as either absolute or relative values. An example demonstrating the difference is indicated in the table below:

Table 37: Absolute and relative coordinates

	First coordinate	Second coordinate	Resultant second coordinate position
Absolute	5 5 5	10 10 10	10 10 10
Relative	5 5 5	10 10 10	15 15 15

Make sure to select “Absolute” or “Relative” before specifying any of the coordinates for the initial and end point as shown in Figure 224. Otherwise, the program will make the adjustments of the new coordinates, which may not produce the expected outcome.

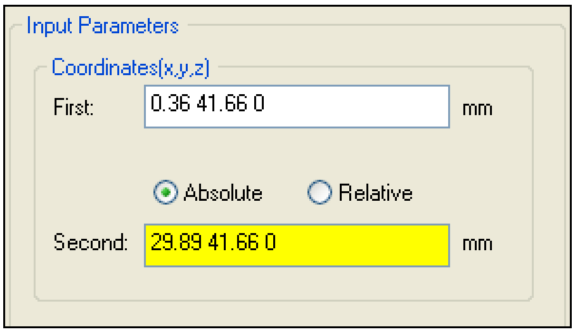


Figure 224: Second coordinate definition

Using filters when modeling

Filters help you select a required model or mesh entity by automatically snapping to the nearest instance of the selected model or mesh entity when you click in the display window.

Filters are available from a drop-down list in the following tools:

- Create nodes tools.
- Create holes tools.
- Nodal mesh tools.
- Set constraints.
- Create curves tools.
- Move/copy tools.
- Edge mesh tools.
- Set loads.
- Create regions tools.
- Create element tools.
- Global mesh tools.

From each tool, an appropriate subset of the filters is available, shown in Table 38. From the drop-down list, select a filter appropriate for your purpose.

Table 38: List of selection filters

Filter	Description
Any item	After selecting this mode, you can click anywhere in the model window, including locations which are not on the model.
Node	When you click on the model, the nearest existing node is selected for you.
Curve	When you click on the model, the nearest existing curve is selected for you.
Center of arc	When you click anywhere on an arc, the center coordinate of the circle described by the arc is selected for you.
End of curve	When you click anywhere on a curve, the point at the nearest end of the curve is selected for you. This option is useful, for example, if you are using **Create Curves** ➡ **Connect**, it is important that you select specific end of the curve rather than the other end.
Middle of curve	When you click anywhere on a curve, the middle point of the curve is selected for you.
Point on curve	When you click anywhere on a curve, the closest coordinate on the curve is selected for you.
Keyboard	Unlike all the other snap modes, input is only accepted from the keyboard. You cannot click directly on the model. You must manually type values into the dialog input box.

Local coordinate systems

MPI allows the creation of Local Coordinate Systems (**LCS**). Multiple LCS's can be defined at any time and set only one to be active at a time. All proceeding model/mesh editing actions and loads/constraints will be based on the active LCS. If the model has results associated, then the LCS will be available for results visualization. The LCS menu options are shown in Figure 225.

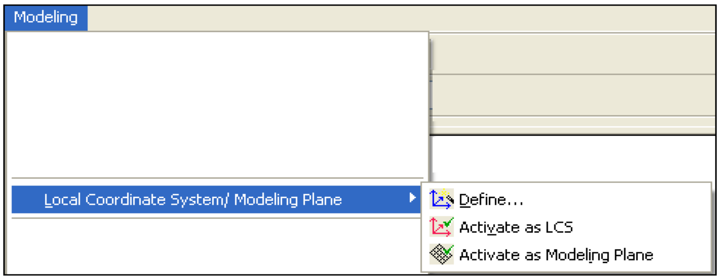


Figure 225: Local coordinate system menu

The LCS menu options allow you to:

- Define (create) a new local coordinate system.
- Activate a LCS so the following geometry will be created relative to the LCS.
- Activate a LCS as a modeling plane so the following geometry will be created in the new XY plane.

To define a local coordinate system you need one to three reference coordinates or nodes in the model, as shown in Figure 226. If a coordinate is not present, the currently active coordinate system is used as a reference.

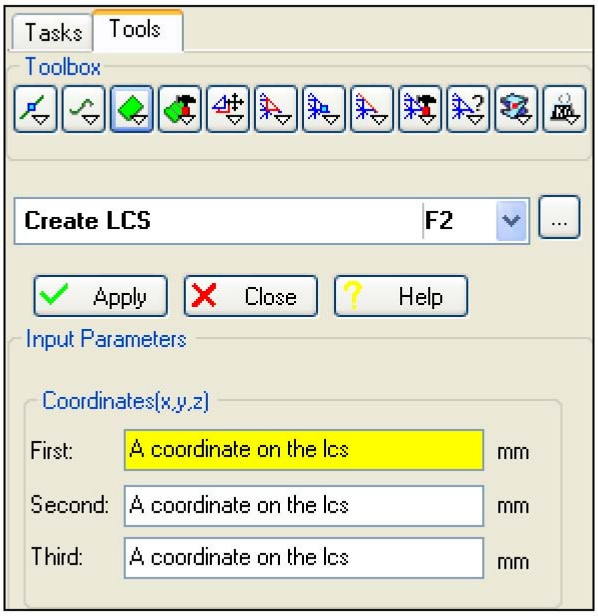


Figure 226: Define local coordinate system input parameters

The three reference locations the X, Y, and Z axis are defined as follows:

Table 39: Coordinates needed to define a local coordinate system

Coordinate	Description
First.	Defines the origin of the LCS.
Second.	Defines the X axis in the LCS.
Third.	Defines the XY plane of the LCS.

The local Z direction is defined to be perpendicular to the XY plane, such that the local coordinate system is "right-handed" as shown in Figure 227.

A right-handed coordinate system is one where the thumb of your right hand represents the Z direction, your extended fingers represent the X direction, and the palm of your hand indicates the local Y direction.

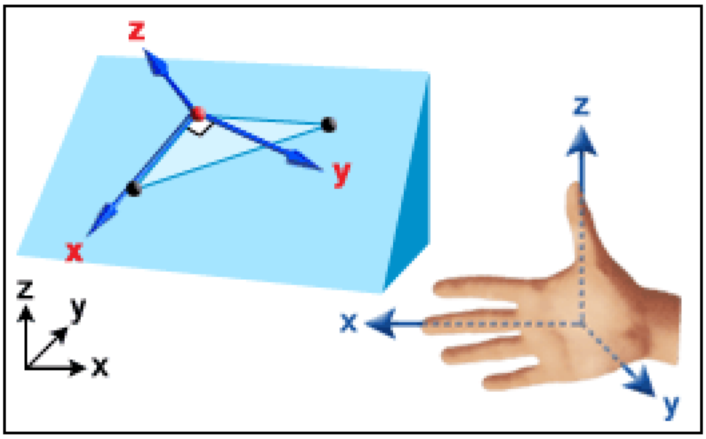


Figure 227: Right hand rule diagram

To define a local coordinate system:

1. Click **Modeling** ➡ **Local Coordinate System/Modeling Plane** ➡ **Define**, to display the Define Local Coordinate System pane.
2. Select the type of model entity you want to snap to when you click in the model display window, this can be done in the Filter drop-down menu at the bottom of the pane.
3. Activate the input box and either enter the required coordinates using the keyboard, or click on the required location in the model display window for each of the 3 Coordinate boxes.
4. Click **Apply** in the Define Local Coordinate System pane to create the local coordinate system.

To activate a local coordinate system:

1. Click the **Select tool** icon  and then click on the local coordinate symbol entity in the model display window.
2. Select **Modeling** ➡ **Local Coordinate System/Modeling Plane** ➡ **Activate** to activate the currently selected local coordinate system.

> An activated LCS is displayed in red.

MPI allows grid-based modeling of a feed system, cooling circuits, and curves. This permits the geometry creation to be quick and easy. The Modeling grid settings specify your preferences for the modeling grid.

To create and use a modeling plane

1. To access the modeling grid specifications click **File** ➡ **Preferences**. The modeling grid specifications are shown in Figure 228:
2. Set the desired units in the System frame if necessary.
3. Enter the required modeling grid size to use.
4. Check Snap to grid if you want the nodes created at the nearest grid point. If unchecked the node will be created exactly where you click.

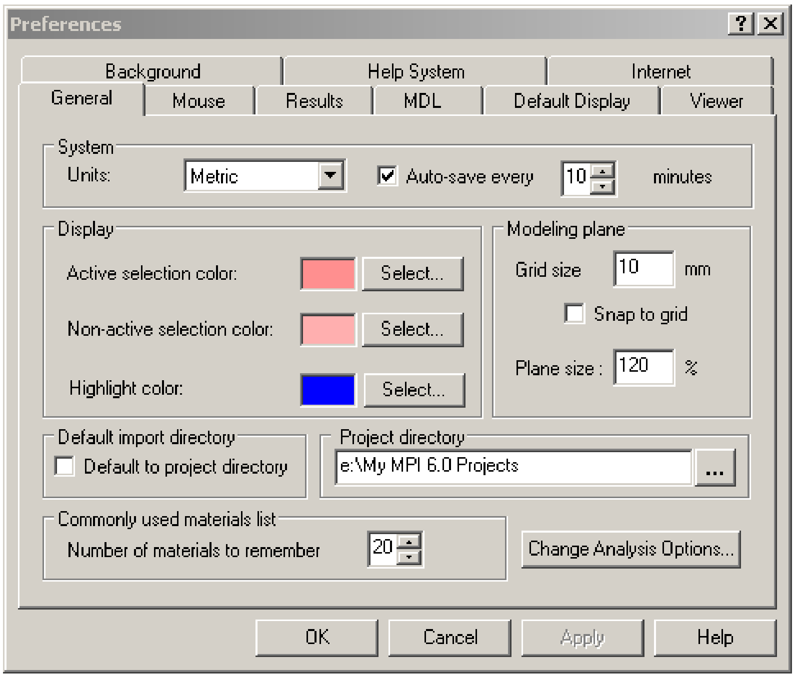


Figure 228: Modeling plane grid preferences dialog

> The grid size specifies the distance between the lines in the modeling grid.

Before you can start creating geometry with a modeling plane, you must define a LCS and activate it as a modeling plane as described in the previous sections.

Modeling using the modeling plane

Below is an example of using a modeling plane to create a hole on an inclined surface.

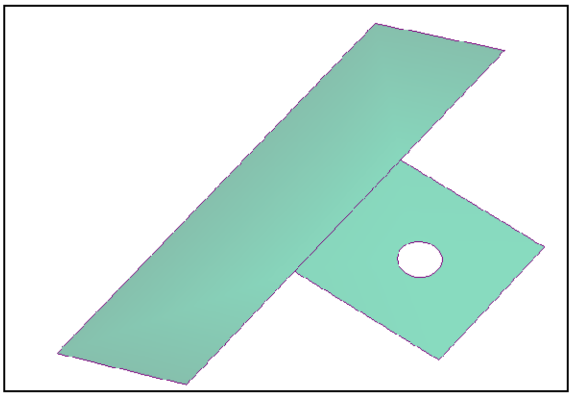


Figure 229: Example of using modeling plane

To create and use a modeling plane

1. Create a convenient LCS. For this specific case, it will be 1/4 from the end of the base part as shown below:

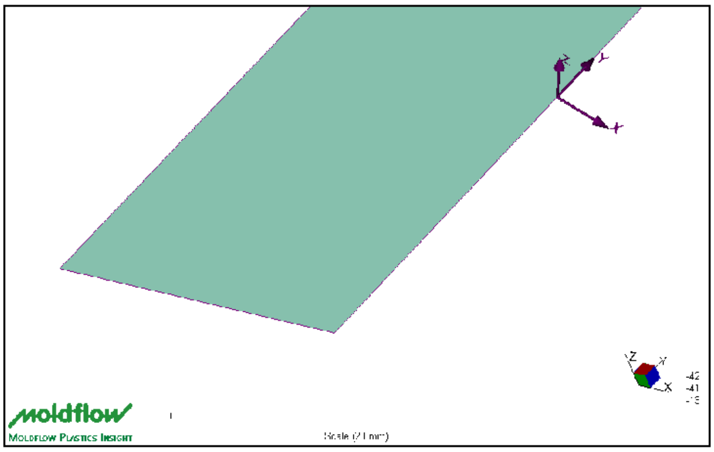


Figure 230: LCS on the side of the part

2. Activate the LCS as a modeling plane.
   - The grid will appear based on the settings in preferences.
   - Anything that you create using the modeling plane will be referenced to the LCS that you set.

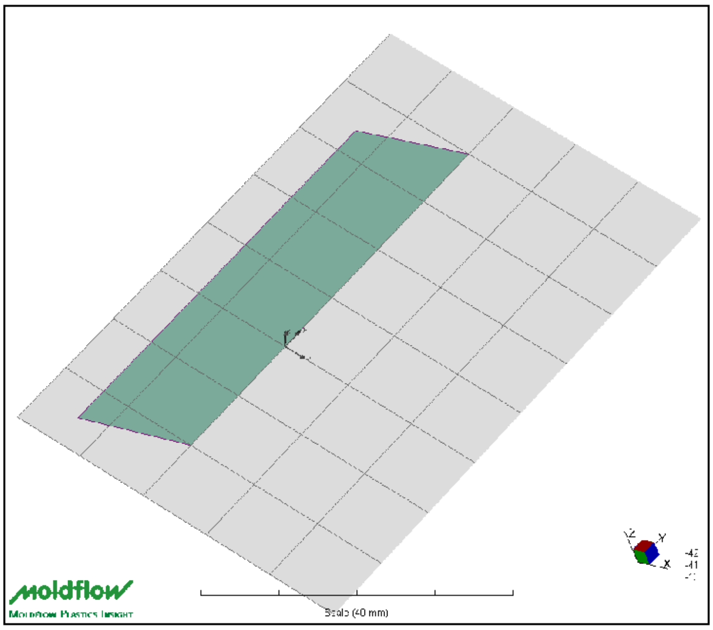


Figure 231: New LCS used as modeling plane

3. Create curves for the incline region using the modeling grid.
4. Define the region using the curves.
5. Create the circle on the inclined region.
    - Use **Modeling Plane** from the filter field in order to have the program select the closest point in the grid.

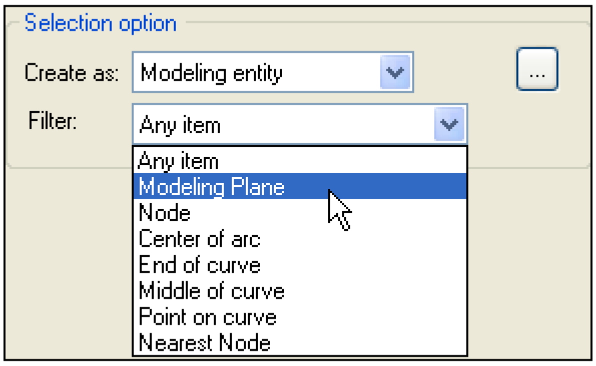


Figure 232: Filter - Modeling Plane

6. Create the hole by going to **Modeling** ➡ **Create Regions** ➡ **Hole by Boundary**. The feature will be as shown below:

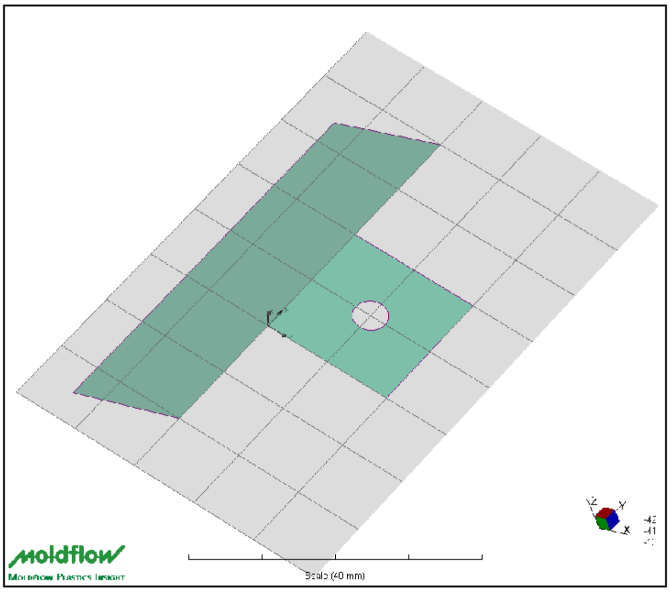


Figure 233: Feature created using modeling plane

Move/Copy Entities

The Move/Copy Entities menu shown in Figure 234, allows you to copy or move select parts of the model.

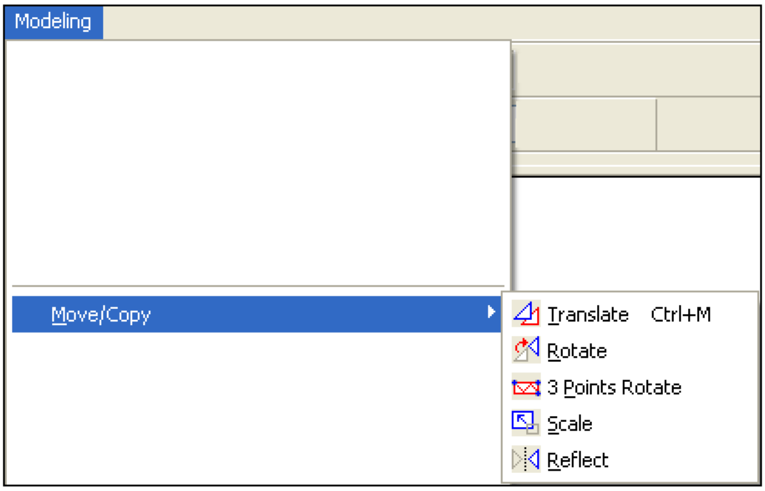


Figure 234: Move/Copy entities menu

The Move option transfers the original entities while the Copy option copies the selected entities and all associated properties.

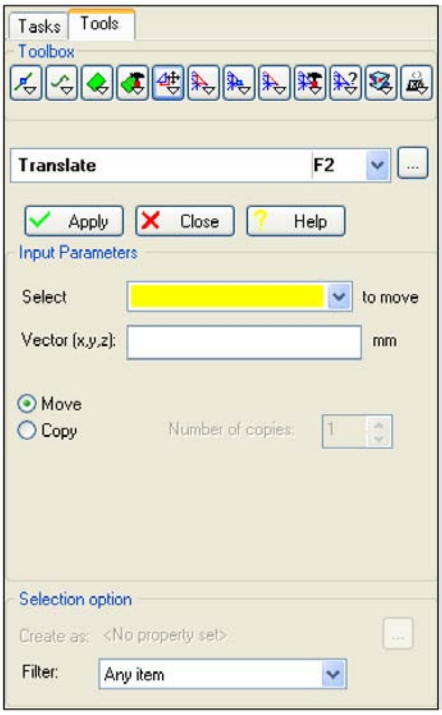


Figure 235: Move/Copy pane

To translate the model

1. Click the Move/Copy icon  in the toolbox, or **Modeling** ➨ **Move/ Copy** ➨ **Translate** to open the Move/Copy (Translate) Entities pane.
2. Select the entities you wish to translate.
3. Specify the vector.
4. Specify whether to move or copy the entities.
5. Click **Apply**.

To rotate the model

1. Click the Move/Copy icon  in the toolbox, or **Modeling** ➨ **Move/ Copy** ➨ **Rotate** to open Move/Copy (Rotate) Entities pane.
2. Select the entities you wish to rotate.
3. Specify the axis around it will rotate.
4. Specify the rotation angle.
5. Specify a reference point. The default is 0.0 0.0 0.0 (i.e. the origin).
6. Specify whether to move or copy the entities.

7. Click **Apply**.

If one of the global axis directions or reference points can't be used, set up a local coordinate system to do the move or copy.

About the 3 point rotate tool

Many parts as they are modeled in the CAD system are not modeled in the same coordinate system as the tool for that part. This complicates the procedure for modeling the runner system and cooling lines inside MPI.

To rectify this you need to move the part into tool position. This often requires the rotation of the part in space using a compound angle. The part can be moved easier using the new 3 Point Rotate tool.

To use the 3 Point Rotate tool:

1. Click the Move/Copy icon in the toolbox, or **Modeling** ➡ **Move/Copy** ➡ **3 Points Rotate** to open Move/Copy (3 Points Rotate) Entities pane.
2. Select the entities to be moved.
3. Enter the coordinates as needed.
4. Click **Apply**.

To scale the model

1. Click the Move/Copy icon in the toolbox, or **Modeling** ➡ **Move/Copy** ➡ **Scale** to open Move/Copy (Scale) Entities pane.
2. Select the entities you wish to scale.
3. Specify the scale factor.
4. Specify a reference point. The default is 0.0 0.0 0.0 (i.e. the origin).
5. Specify whether to move or copy the entities.
6. Click **Apply**.

To reflect the model

1. Click the Move/Copy icon in the toolbox, or **Modeling** ➡ **Move/Copy** ➡ **Reflect** to open Move/Copy (Reflect) Entities pane.
2. Select the entities you wish to reflect.
3. Click the mirror plane.
4. Specify a reference point. The default is 0.0 0.0 0.0.
5. Specify whether to move or copy the entities.
   - **Optional**: If you select copy, select Attempt connection to existing model to allow MPI to connect the new entities to the existing model.

6. Click **Apply**.

What You've Learned

Basic concepts used in CAE software specifically inside MPI. The entities that can be imported or created inside MPI are:

- Mesh (elements and nodes).
- Curves.
- Regions and surfaces.

The modeling menu allows you to:

- Create:
    - Nodes.
    - Curves.
    - Regions.
    - Local coordinate systems (LCS).
- Use local coordinate systems as a modeling plane.
- Move or Copy entities.
- Query entities.

Most of the entries for the modeling tools menus allow you to:

- Specify coordinates in different ways.
- Use relative or absolute coordinates.
- Use filters to aid the selection of the entities.

An entity can be created or modified using different tools or a combination of them.

CHAPTER 11

Introduction to Moldflow Magics STL Expert

Aim

Learn how Moldflow Magics STL Expert can be used to fix & optimized the original CAD file.

Why do it

When importing a model inside MPI previously optimized by Moldflow Magics STL Expert the cleanup time process inside MPI is reduced.

Overview

In this chapter, you will be introduced and practice using Moldflow Magics STL Expert. You will learn about:

- Program overview.
- Supported Models.
- Licensing & Hardware Support.

You will practice with the program features:

- STL Fix Wizard.
- Manual Fixing Tools.
- Optimization Tools.
- Measurement Tools.

Theory and Concepts - Introduction to Moldflow Magics STL Expert

Moldflow Magics STL Expert is a powerful tool for viewing, measuring, correcting and optimizing Stereolithography (STL) and solid surface models imported from popular 3D CAD/CAM systems in preparation for analysis using Moldflow Plastics Advisers (MPA) or Moldflow Plastics Insight (MPI). Moldflow Magics STL Expert can be used to quickly validate STL models and resolve any identified problems.

Moldflow Magics STL Expert is a Windows based program, which makes it easy to use and very flexible due to the proliferation of PC based software. Pull-down menus and icons enable you to handle files and parts in a short period of time. The program is based on STL-format (standard triangulation language) files. STL files describe surfaces as a collection of triangles, which makes them ideal for use in Rapid Prototyping. Most CAD systems can export STL files, making Moldflow Magics STL Expert compatible with all major CAD formats, for example, IGES, VDA, CATIA, VRML, and Unigraphics.

During the translation of 3D data from one system to another many problems can occur, even while using the highest quality data. The Moldflow CAD connectivity tools were developed to facilitate the file translation and address the challenges common to plastic part design when transferring from CAD to CAE systems. Depending on nature of the CAD models used, the user may use one or more of the following products:

- Moldflow Design Link (MDL).
- Moldflow Magics STL Expert.
- Moldflow CAD Doctor.

Corrected and optimized STL models can be output in Moldflow's proprietary UDM (Unified Data Model) format, which can then be imported into MPA 7.1 and MPI 5.0 Revision 1 or higher.

Loading a part

This command opens a dialog for loading a part into Moldflow Magics STL Expert. To access this dialog, click **File ➡ Load Part**, or use **Ctrl+L**.

Moldflow Magics STL Expert 1.0 has several demo files available for use after installation. A Thumbnail picture of any file can be viewed by using the preview feature in the **Open** dialog. You can load more than one part at a time by pressing the **Control** key, while selecting the parts with the left mouse button.

> You can also use the Load Part function to load MGX files (MGX files are compressed STL files using Moldflow Magics STL Expert or STLZip).

Unloading a part

This command removes only selected parts. If several parts are selected, these parts are removed all at the same time. The **Unload** command does not affect any platform settings. To access this command, click **File ➡ Unload Part**, or use **Ctrl+U**.

Before unloading the part(s), a prompt will appear asking you to save the part(s) if they have been changed. Either click **Yes, No**, or **No To All** to this prompt.

Unloading all files

This command unloads all parts that you currently have open. A prompt appears asking you to save your file(s) before performing this action. If you select to save, it will be in ***.pff** file format. To access this command, click **File ➡ New Platform**.

Using this command will completely empty the **Undo** folder.

Unit conversion

This command converts the units display of selected parts from metric to English, and English to metric, while retaining the original part dimensions. To access this command, click **Edit ➡ Unit Conversion**.

This function is required because the STL-format does not contain information concerning the measurement system used, and Moldflow Magics STL Expert cannot determine the difference between one millimeter and one inch.

When working with multiple parts of differing units, the unit conversion must be used. When loading parts, Moldflow Magics STL Expert warns you if the part has units different from a part that is already loaded, or from the setting that is active in the pull-down menu. Either click **Yes** or **No** to this prompt when it appears.

Be careful when loading parts to ensure that the parts are in the required units.

Importing files

To access this dialog, click **File ➡ Import**.

Moldflow Magics STL Expert allows you to import various CAD formats such as DXF (with 3D faces), VRML, IGES, and more. DXF and VRML import capabilities are included with all installations; others such as IGES and CATIA are available as add-on modules.

To import a CAD file, click **File ➡ Import**, select the type of file you want to import, and ensure that the only check box selected is **Make all Entities visible**. Importing files in this manner allows you to see where there are any imperfections that might need to be fixed.

An important aspect of importing files is **Accuracy**. The accuracy determines how closely the triangulated part resembles the original, and how many triangles are used. The smaller the number, the more triangles used for the part description.

Help files

The access the Help, click **Help ➡ Magics STL Expert Help**, or click **F1**.

The online Help provided with Moldflow Magics STL Expert is in HTML Help format. This makes them easy to use because they act as a web type help page.

There are several navigational functions in the Help that allow you to search for and locate the information you want to find. The main functions that can be used are the Contents, Index, and Search tabs.

Use the **Contents** tab like you would any other table of contents, to browse through the Help content and click on a link of interest. You can use the **Index** tab to search for the first letters or words of an area of the program you need assistance with. When you have found the term, click on it. You can also use the **Search** tab to search on particular words or phrases. Every online Help topic that contains the word(s) you searched on will be listed in the search results.

Getting Started: Visualization, Looking Inside the Model (Sections) & Measuring

Moldflow Magics STL Expert offers several options for visualizing and manipulating STL files. These visualization and manipulation tools are:

Mouse Manipulation Controls

- **Rotate**: Right mouse button.
- **Pan**: Right mouse button+ Shift.
- **Zoom In & Out:**
  - Right mouse button + Ctrl.
  - Mouse wheel.

View Toolsheet

The View Toolsheet, shown in Figure 266, aids in the visualization of the model. The Mode tab options are: Shade, Wire, Shade & Wire, Triangle view, and Box.

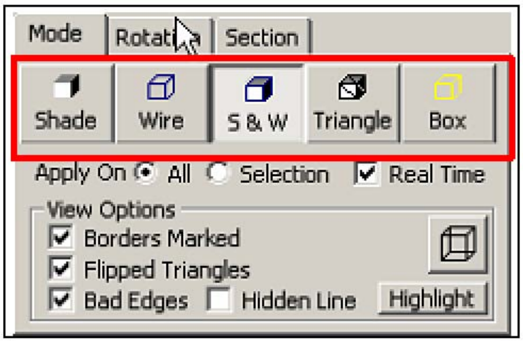


Figure 266: View Toolsheet - Mode options

The View Toolsheet helps visualizing different errors, shown in Figure 267.

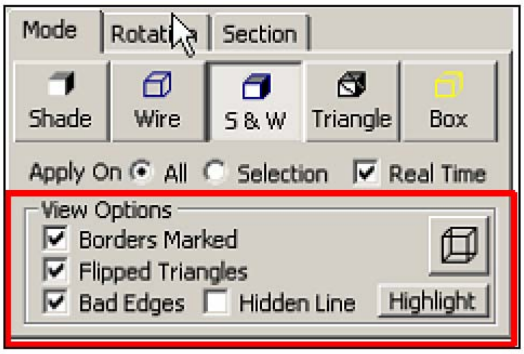


Figure 267: View Toolsheet - View options

The Section tab can be used to measure, check for errors and to look inside the model to get a better understanding of the geometry. Figure 268 displays the options of the Section tab.

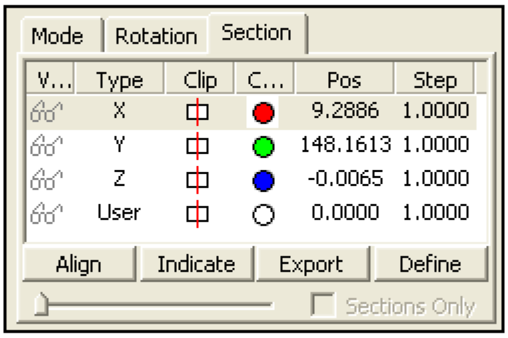


Figure 268: Section tab

Important items on this toolbar are the following:

- **Visible Column** (first): Indicates which section you want to use (only X).
- **Clip Column**: Indicates what you want to see (only front, only back, all).
- **Indicate button**: The position of the section can be pointed on the part.

- **Slider bar**: You can take parallel sections by sliding the slider in the slide bar. Move the selected section through the shell model by moving the slider.

Measurement Toolsheet

The measurement toolsheet has different tabs. You can measure **distances, radii** (or **diameters)** and **angles.** You can also display some coordinate **information** of different features in the part: a point, a line, a triangle, a cylinder, a circle and a sphere. The options available update as you change the tab from Distance to Info displayed in Figure 269.

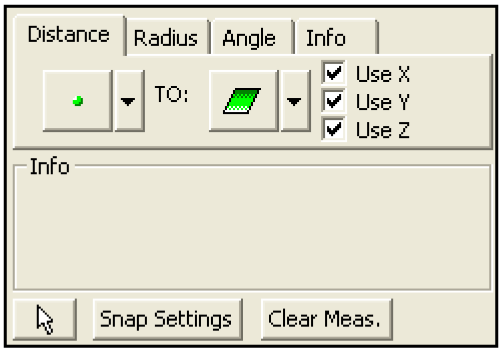


Figure 269: Measure Toolsheet

To use the Measure Toolsheet you need to choose which entities you will select on your part:

7. Select entity 1 using the left mouse button.
8. Select entity 2 using the left mouse button.
9. Position measurement using the left mouse button.

STL Optimization tools

Moldflow Magics STL Expert has optimization tools. These tools are located in the Mark Toolbar, see Figure 270.

- De-Featuring - mark & delete features that you don't need.
- Triangle Reduction - same geometry, less triangles.
- Remesh - same geometry, triangles of a higher quality.

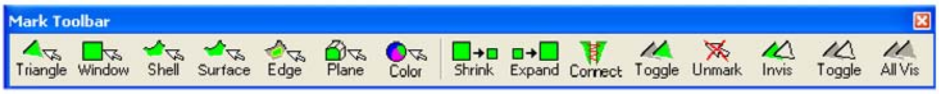


Figure 270: Mark Toolbar - Optimization tools

Supported Models

You can import the following formats directly into Moldflow Magics STL Expert:

- STL.

- IGES.
- MGX (Materialise compressed STL format).

Moldflow Design Link (MDL) is an add-on product which provides faster and easier access to CAD data allowing users to import leading solid geometry formats directly into Moldflow Plastics Advisers (MPA), Moldflow Plastics Insight (MPI) or into Moldflow Magics STL Expert. MDL enable users to import files such as:

- Parasolid.
- SolidWorks.
- Pro/ENGINEER.
- CATIA V5.
- STEP.

Customers with MDL can import IGES models either directly or via MDL. Importing IGES models via MDL instead of directly was found to generate better export quality models.

Licensing & Hardware Support

Moldflow Product Security (MPS) is required to run all Moldflow software. It allows your purchased Moldflow software to run on a standalone pc (node-locked license) or over a network (floating license), or a combination of the two. If you have other Moldflow software installed, this is likely to already be running. Check with your network administrator if you are unsure.

The supported hardware for Moldflow Magics STL Expert is Intel Pentium/Xeon-based Windows XP/2000.

What You've Learned

Moldflow Magics STL Expert is powerful tool for:

- Viewing.
- Measuring.
- Correcting.
- Optimizing Stereolithography (STL) and solid surface models.

Such models are imported from popular 3D CAD/CAM systems in preparation for analysis using Moldflow Plastics Advisers (MPA) or Moldflow Plastics Insight (MPI). Moldflow Magics STL Expert can be used to quickly validate STL models and resolve any identified problems.

Moldflow Design Link (MDL) can be used to expand the formats capable to be read inside Moldflow Magics STL Expert.

There are many ways that commands can be accessed using menus, shortcut keys and toolbars. Most commands have several methods to activate them.

The View Toolsheet aids in the visualization of the model. The Mode tab options are:

- Shade.
- Wire.
- Shade & Wire.
- Triangle view.
- Box.

The measurement toolsheet has different tabs. You can measure:

- Distances.
- Radii (or diameters).
- Angles.

Moldflow Magics STL Expert has optimization tools such as:

- De-Featuring.
- Triangle Reduction.
- Remesh.

CHAPTER 12

Material Searching and Comparing

Aim

The aim is to learn how to search for and compare different thermoplastic materials in the material database.

Why do it

To run an analysis in MPI, you need to select a material. The choice of material may be predetermined, or a material can be selected based on the results of a database search. To help in the selection of the material, one material can be compared to one or more materials to see what the variation between the materials will be.

Overview

There are several ways a material can be selected for an analysis, depending on the available information. If the manufacturer and trade name are known, you can simply select it from a list. If the material to be used is not known, a material search can be performed to find a suitable material. Having selected a material, you can compare it to other materials, or view a report on the quality of the material data.

Note the answers are based on MPI 6.0 build 05495. If a different build of software is used, the answers may be slightly different.

Theory and Concepts - Material Searching and Comparing

Select material dialog

The **Select Material** dialog, shown in Figure 288, allows you to select the material to be used in an analysis. It is your access point to the extensive material database supplied with MPI. You can use this dialog to plot the properties of a material, search for a specific material, select a material, and add the material to a commonly used materials list. The select material dialog is opened by clicking the **Material** icon in the Study Tasks list, or by the menu, **Analysis** ➡ **Select material**.

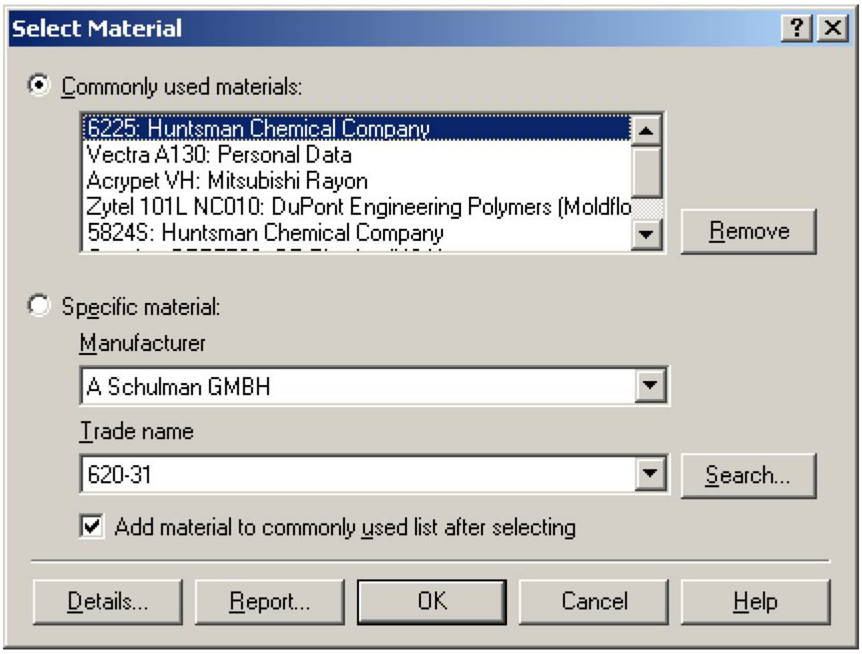


Figure 288: Select Material dialog

Commonly used materials

The **Commonly used materials** area lists the materials that you have previously selected. By default, up to 20 of your most recently selected materials are displayed in the list. You can increase or decrease this limit in the program preferences. Every time you select a material in the lower half of the dialog, and the **Add material to commonly used list after selecting** check box is selected when you close the dialog, the material will automatically be added to the Commonly used materials list. This eliminates the need to repeatedly search the database for the materials that you use more frequently.

Specific material

The **Specific material** area contains a list of every entry in the material database (currently over 7600 materials), sorted by **Manufacturer**, and then **Trade Name**. To locate an entry in the manufacturer or trade name drop-down list:

- Click the down-arrow on the right-hand end of the list control and then use the scroll bar to scroll through the alphabetically sorted list of names, or,
- Click the drop-down list to select it, type the first few letters of the manufacturer or trade name, then use the arrow keys or pull down the list to narrow down your search.

Searching

The **Search Criteria** dialog is accessed via the **Search** button on the Select Material dialog. It allows you to search the thermoplastics database according to defined filter criteria. For example, you might search for all plastic materials whose family abbreviation includes **TP**, or whose filler description is **glass**, as shown in Figure 289.

After defining a set of search criteria, you can save it using the **Save** tool  in the top right corner of the dialog, and reopen it at a later time using the **Open** tool .

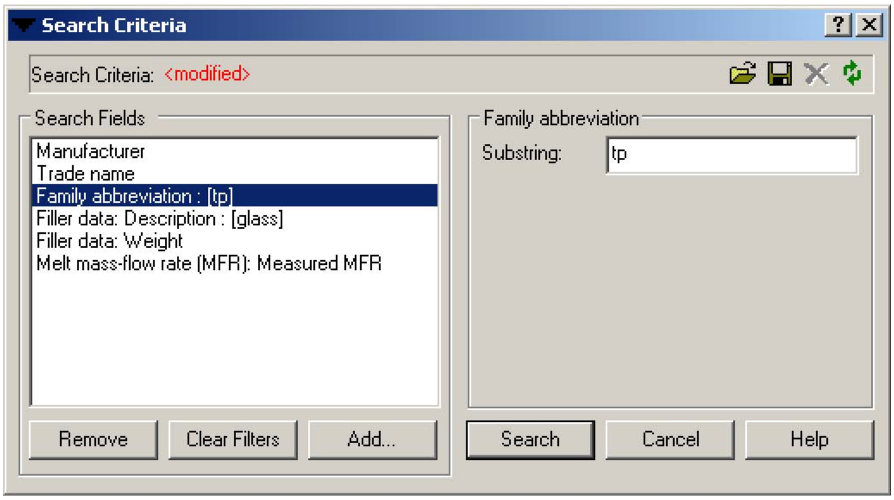


Figure 289: Search Criteria dialog

Search Fields

The **Search Fields** area provides the following default list of search fields:

- Manufacturer.
- Trade name.
- Family Abbreviation.
- Filler data: Description.
- Filler data: Weight.

- Melt mass-flow rate (MFR): Measured MFR.

You can add or remove search fields using the **Add** and **Remove** buttons below the list. The **Add** button opens the Add Search Fields dialog, which lists most of the data items stored in the materials database. New search fields are added to the end of the existing Search Fields list. The **Remove** button allows you to delete the currently selected search field from the list.

The **Clear Filters** button is used to remove any filter values (substrings, minimum, maximum, etc.) that you have entered for the search fields.

Commonly used search fields

In addition to the default search fields listed above, there are many other search fields that can be used. Three of the commonly used fields include:

Moldflow viscosity index

This is a number in the form of **VI(240)0152** where the number in the parentheses is the melt temperature in Celsius, and the 4 digit number to the right of the temperature is the viscosity in Pa *sec., measured at a shear rate of 1000 1/sec.

This is a good single point value for comparing the relative viscosity of one material to another. The higher the number, the higher the material viscosity. The shear rate value of 1000 1/sec. is important as it is in a typical range that is seen in injection molding. Every material on the database has a 4 digit viscosity index. The melt temperature for a given material family will always be the same, so the search results can be sorted by viscosity index. The Moldflow viscosity index is an excellent tool to use for the initial assessment of a material.

Data source

The data source indicates who tested the material and provided the results to Moldflow. The source falls into 3 categories:

- Moldflow.
- Manufacturer.
- Other.

All data in the material database goes through a quality assurance procedure to check whether that data is reasonable or not. However, for the materials that Moldflow did not test, we have to rely mostly on the source of the data to ensure the data is good.

In many cases, the materials that Moldflow test are also mold-verified. This means that after all the material properties are measured, it is compared with actual moldings to ensure the pressure in the simulation is similar to what was measured on the press.

The less information you have about a material, the more likely the results may be negatively influenced by the quality of the material data. One thing you can do to validate the material data is to run an analysis with a different grade material of the same type and filler content, and with a similar viscosity index. If the second material is Moldflow tested, and the results are similar, you can feel more confident that the original material data is good.

Corrected residual in-mold stress (CRIMS) model …

Many analysis sequences today will end up including a warpage analysis. A warpage analysis can be performed with any material on the database, but the most accurate warpage results are achieved for materials that have been shrinkage tested. The easiest way to find a material that has shrinkage data is to examine the database field in which that data is stored. If in the search results, the field is empty, this indicates that no shrinkage data is available. After the first search, you can sort by this field to find the minimum and maximum shrinkage values. This can be used to limit your search to only materials that have shrinkage data.

Substring

The Substring text box located on the right-hand side of the Search Criteria dialog allows you to specify the text or numerical characters you wish to search on in the currently selected search field.

For example, if you select **Family Abbreviation** as your search criteria, and you type **TP** as the substring, then the search will find materials with the following Family Abbreviations;

- ETPU
- PP+TPO
- TPE
- PP+TPE
- SEBS+TPE
- TPE-S
- TPEE
- TPI
- TPO
- TPR
- TPU
- TPU+PC
- TPV
- TPX

The search engine will find any occurrence of the substring in the search field.

If you are not sure in what format the data for a particular search field is, run a preliminary search with the substring text box empty. This will output all values for that field.

Saving and loading search criteria

Once you have defined a search criterion, it can be saved for quick retrieval at a later time. This saves you having to constantly reconstruct the same set of search criteria. Click the diskette icon to open the Save Search Criteria dialog as shown in Figure 290.

Click the folder icon to open a previously saved set of criteria. After loading a saved set of criteria, you can easily return to the default criteria settings by clicking the **Reset To Default Search Criteria** icon.

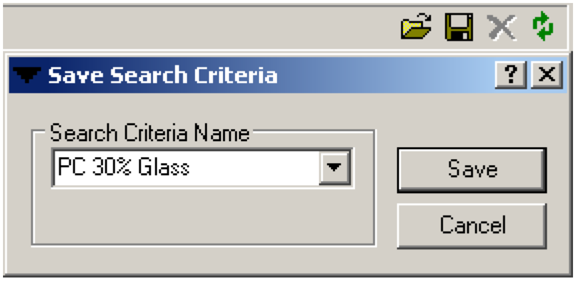


Figure 290: Save search criteria dialog

You can use this feature to save an alternate generic search criteria list to the default one supplied with MPI. It will save time because you will not need to re-build the list over and over again. An example generic search criteria list might be:

- Manufacturer.
- Trade name.
- Family Abbreviation.
- Moldflow Viscosity Index.
- Data source.
- Filler data: Description.
- Filler data: Weight.
- Corrected residual in-mold stress (CRIMS) model …

Reviewing search criteria results

Search results are displayed in the **Select Thermoplastics material** dialog. Clicking on a column heading sorts that column. In Figure 291, the Trade name column has been clicked. It has a little up arrow indicating the sort is ascending. If you click the column heading again, the results will be redisplayed in descending order. You can change the order of a column by dragging and dropping the column heading. This may be useful in helping narrow down the list of materials you are interested in.

You can export the search results to a tab delimited text file by clicking the **Export** button and entering a file name.

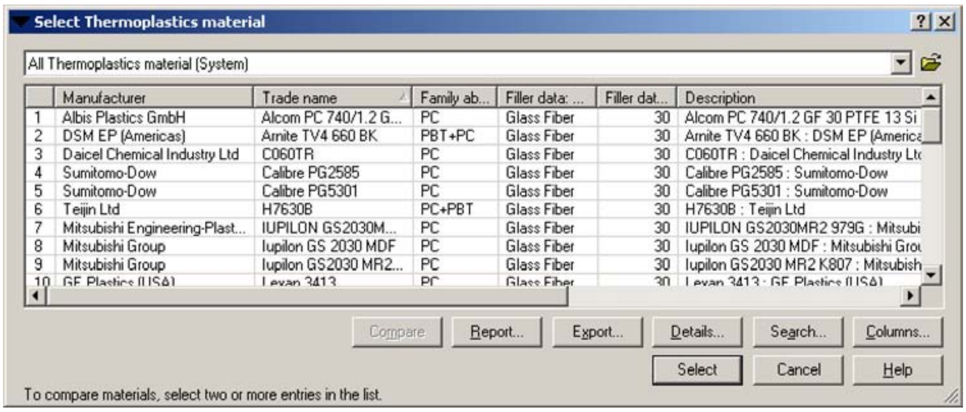


Figure 291: Search results

Organizing database columns

Clicking the **Columns** button on the Select Thermoplastics material dialog will open the **Columns** dialog as shown in Figure 292. You can use this dialog to adjust the order of the columns, and individually display or hide each of the available data fields. By default, only those data fields in the Search Fields list on the Search Criteria dialog will appear in the output of the search. You can add additional fields that were not used as search criteria using the Columns dialog.

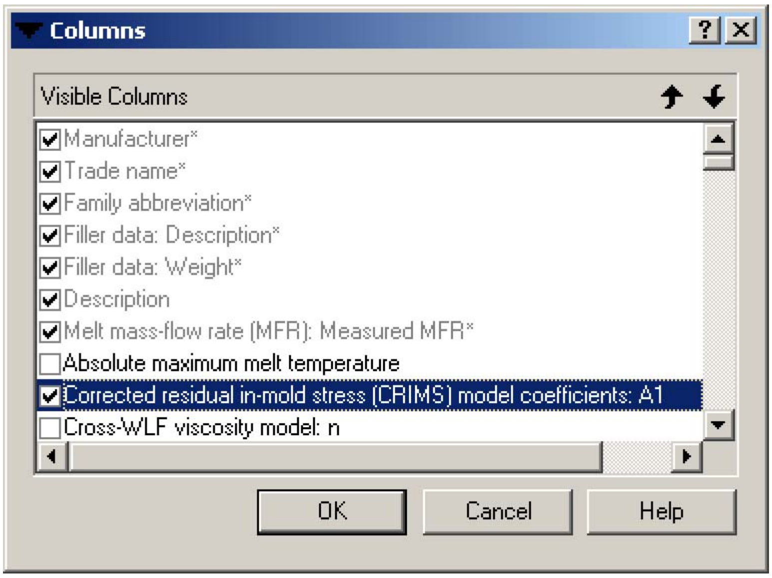


Figure 292: Columns dialog

Material details

The **Thermoplastics material** dialog shown in Figure 293, can be accessed by:

- Right-clicking the **Material** icon  in the Study Tasks pane.
- Clicking the **Details** button on the Select material dialog, shown in Figure 288 on page 405.
- Clicking the **Details** button on the **Select Thermoplastic material** dialog, shown in Figure 291 on page 409.

The **Thermoplastics material** dialog has eight tabs. The tabs have detailed information in the following categories:

- Description.
- Rheological Properties.
- PVT Properties.
- Shrinkage Properties.
- Recommended Processing.
- Thermal Properties.
- Mechanical Properties.
- Filler Properties.

If a property name is shown in red, it indicates that the property has not been tested for that specific material, and that generic data is being used. Generic data can come from a similar grade of material, or in some cases, a reference material. Not all properties can be assigned generic data.

Most tabs provide buttons to access further information, for example, to view testing information, or to plot the data.

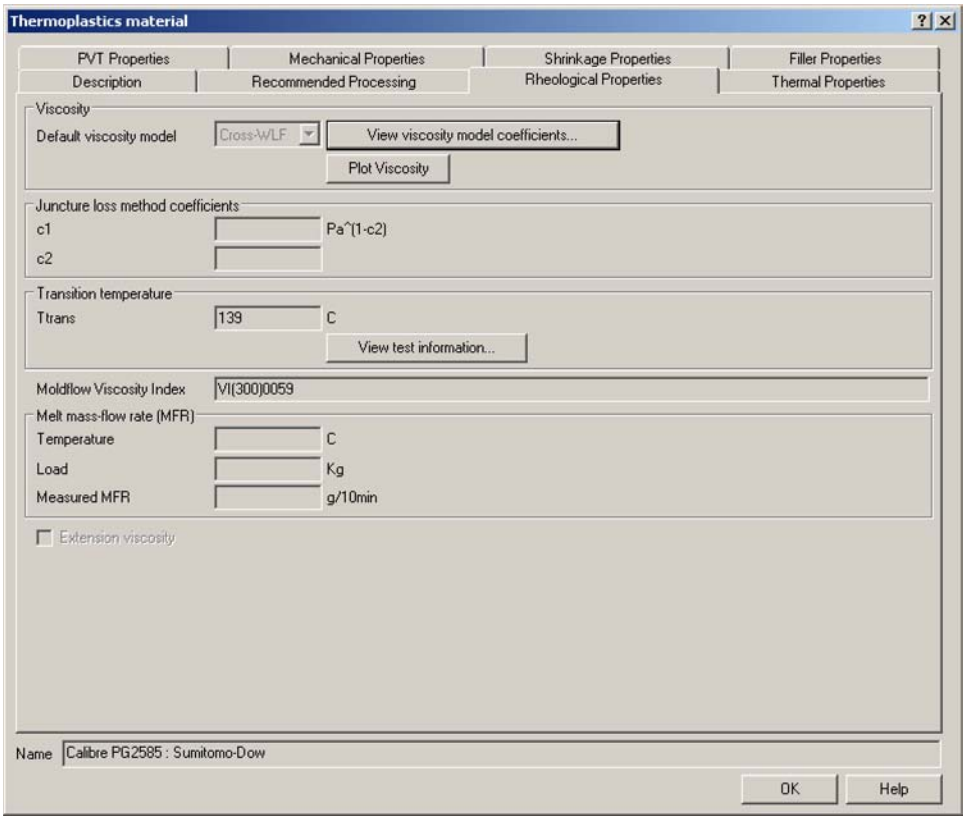


Figure 293: Thermoplastics material dialog

When you click the **Plot Viscosity** button shown in Figure 293, the viscosity data for the material is plotted in a separate window, as shown in Figure 294.

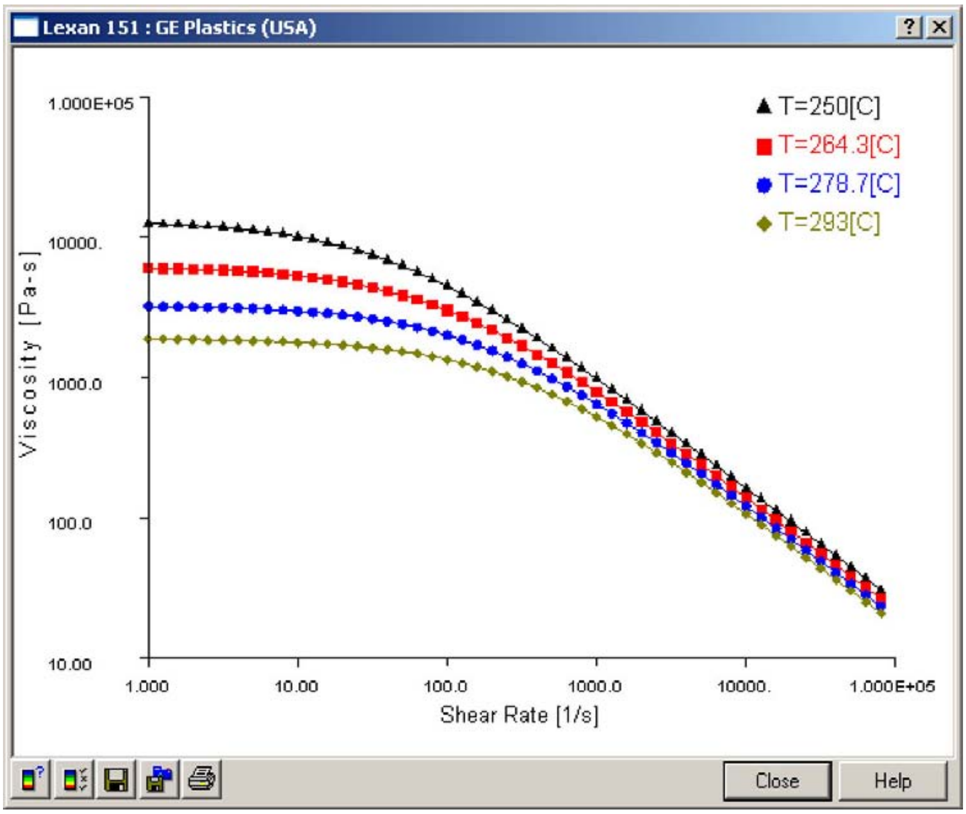


Figure 294: Viscosity plot

Material report

The **Material Data Method Report** dialog, shown in Figure 295, allows you to view material testing information on the selected material. The report indicates the type of testing that was done. This report is important to review when picking materials to use for an analysis. You would like to run an analysis with the highest quality material data possible. This report will help you determine the quality of the data. When you have a choice in materials to use, picking a material with the highest quality of testing methods will give you more confidence that the analysis will be good.

You can view the Material Data Method Report by:

- Right-clicking on the material in the **Study Tasks** pane and selecting **Report**.
- Clicking the **Report** button on the **Select Thermoplastics material** dialog.
- Clicking the **Report** button on the **Select Material** Dialog.

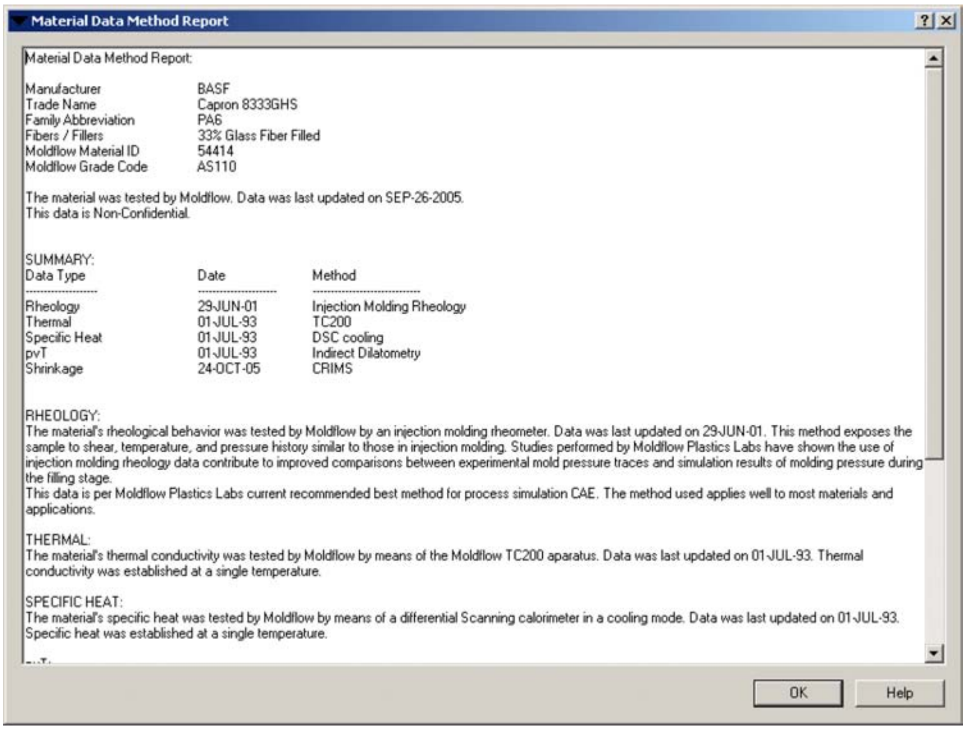


Figure 295: Material data method report

Compare materials

The **Compare materials** utility creates a report of two or more materials that lists the data and test methods. You can this utility by:

- Right-clicking on the material in the **Study Tasks** pane and selecting **Compare With**, as shown in Figure 296.
- Selecting two more materials on the **Select Thermoplastics material** dialog and clicking the **Compare** button Figure 291 on page 409.

Figure 296: Compare materials

When using the context menu, the **Select Materials To Compare With** dialog opens and allows you to pick another material to compare with the material selected in the Study Tasks list shown in Figure 297. Select a material from the dialog by scrolling through the list or conducting a search.

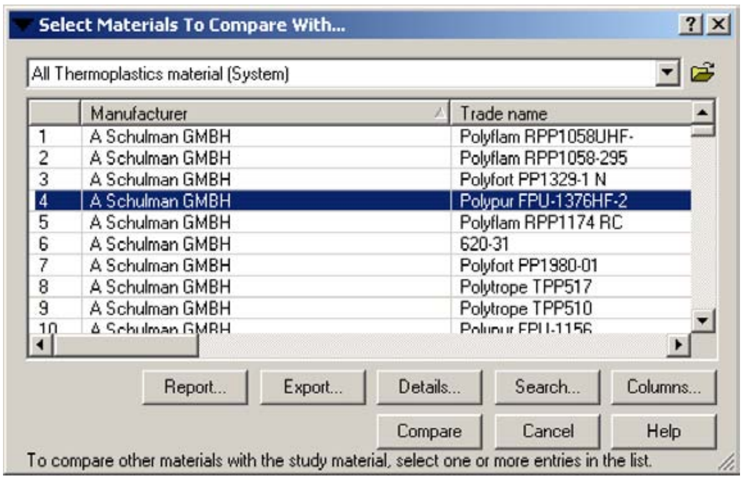
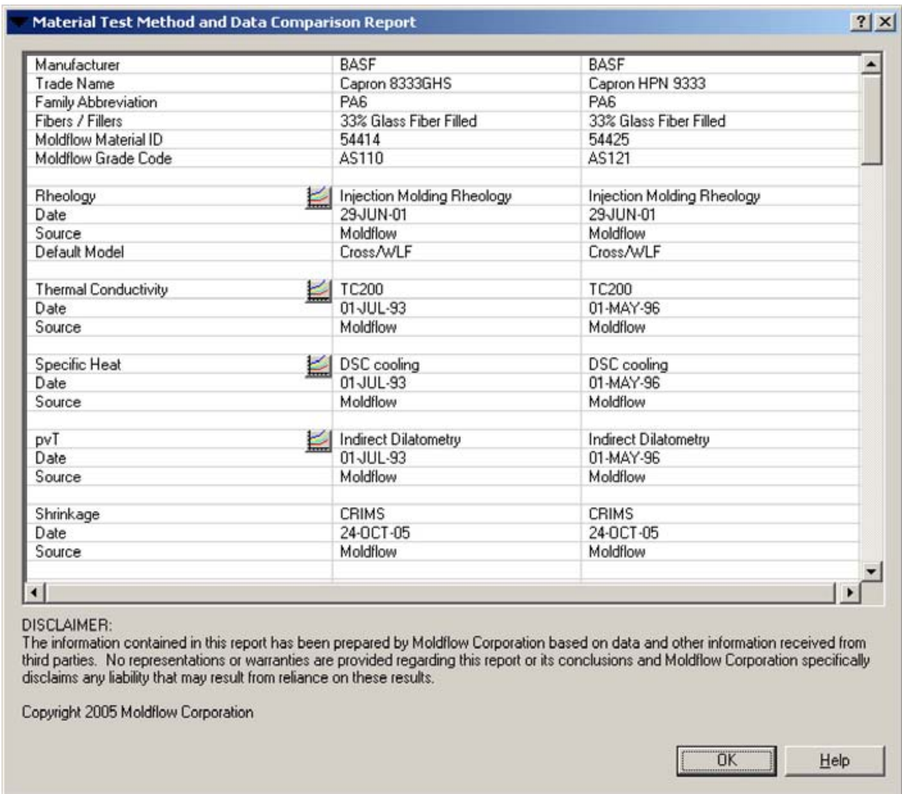


Figure 297: Select materials to compare with dialog

Once you have selected two or more materials, click **Compare** to display the dialog shown in Figure 298. The dialog lists all the types of data, and shows the values for each material. In some cases, for example viscosity and PVT data, there is an option to plot the data to compare it.

This utility can be used as an aid to understanding how one material behaves compared to another. If is also useful for diagnosing analysis problems. If a flow analysis result is not what you expected for some new material, you can compare that material to one you know better to see if you can find a material explanation for the flow results you have seen.



Material Test Method and Data Comparison Report

Manufacturer	BASF	BASF
Trade Name	Capron 8333GHS	Capron HPN 9333
Family Abbreviation	PA6	PA6
Fibers / Fillers	33% Glass Fiber Filled	33% Glass Fiber Filled
Moldflow Material ID	54414	54425
Moldflow Grade Code	AS110	AS121
Rheology	Injection Molding Rheology	Injection Molding Rheology
Date	29-JUN-01	29-JUN-01
Source	Moldflow	Moldflow
Default Model	Cross/WLF	Cross/WLF
Thermal Conductivity	TC200	TC200
Date	01-JUL-93	01-MAY-96
Source	Moldflow	Moldflow
Specific Heat	DSC cooling	DSC cooling
Date	01-JUL-93	01-MAY-96
Source	Moldflow	Moldflow
pvT	Indirect Dilatometry	Indirect Dilatometry
Date	01-JUL-93	01-MAY-96
Source	Moldflow	Moldflow
Shrinkage	CRIMS	CRIMS
Date	24-OCT-05	24-OCT-05
Source	Moldflow	Moldflow

DISCLAIMER:
The information contained in this report has been prepared by Moldflow Corporation based on data and other information received from third parties. No representations or warranties are provided regarding this report or its conclusions and Moldflow Corporation specifically disclaims any liability that may result from reliance on these results.

Copyright 2005 Moldflow Corporation

OK Help

Figure 298: Material test method and data comparison report

Required material data

Table 40 below summarizes the material data needed for various types of analysis.

Table 40: Material data stored in the Thermoplastics Material Database

	Flow	Fill	Cool	Flow + Warp	Stress	Molding Window	Gate Location
Description	info	info		info	info	info	info
Processing temperatures	✓	✓	✓	✓		✓	✓
Ejection temperature	✓	✓	✓	✓		✓	✓
Viscosity model	✓	✓		✓		✓	✓
Juncture loss	opt	opt		opt		opt	opt
Extensional viscosity model (3D)	opt	opt		opt		opt	opt
Transition temperature	✓	✓	✓	✓		✓	✓
Moldflow viscosity index							
Melt flow rate							

Table 40: Material data stored in the Thermoplastics Material Database

	Flow	Fill	Cool	Flow + Warp	Stress	Molding Window	Gate Location
Specific heat (single pt)	✓	✓	✓	✓		✓	✓
Specific heat (tabulated)	opt	opt	opt	opt		opt	opt
Thermal cond (single pt.)	✓	✓	✓	✓		✓	✓
Thermal cond (tabulated)	opt	opt	opt	opt		opt	opt
Solid density from PVT	✓	✓		✓	✓		
Melt density from PVT	✓	✓		✓			
PVT data	✓	✓		✓		✓	✓
Mechanical properties	info	info		✓	✓		
CTE data	info	info		✓	✓		
Stress at yield							
Strain at Break							
Shrinkage model	info	info		✓			
Observed shrinkage							
Filler data	✓	✓		✓	✓		

Table 41: Key for Table 40 above

✓	Required and used by the solvers.
info	Required but used only for restart functions or information field.
opt	Not required, used if provided.
	Not required, not used by solvers (information only).

What you've learned

Picking a material to be used for an analysis is done from the **Select Material dialog**. On this dialog, there are two places materials can be chosen from:

- Commonly used materials.
- Specific material.

In the commonly used materials section, the 20 last materials chosen are listed. The number stored in this list, is defined in Preferences.

In the Specific material area, a material can be chosen by:

- Selecting the Manufacturer then Trade name from pull down lists.
- Searching.

When searching, the Search Criteria dialog is opened. On this dialog, search fields are defined. Commonly used search fields include:

- Manufacturer.
- Trade name.
- Family abbreviation.
- Moldflow Viscosity Index.
- Data source.
- Corrected residual in-mold stress (CRIMS) model …
- Filler data.

For any search field, a substring can be entered to define the data to be searched. For example, **Moldflow** can be entered in the Data source field to find all materials that were tested by Moldflow. Search criteria can be saved for future used.

Once a search has been done, a dialog called **Select Thermoplastics material** is opened. Each column represents one of the search fields defined. The columns can be sorted and rearranged to facilitate the searching.

From the Select thermoplastics material dialog, the following can be done:

- Material details, this allows you to view all the data stored for a material.
- Material report, this opens a report summarizing the testing methods used to test the material.
- Compare materials, this allows you to compare the properties of two or more materials.

The material database stores data required for an analysis, and some minor general information.

Note the answers are based on MPI 6.0 build 05495. If a different build of software is used, the answers may be slightly different.

CHAPTER 13

Gate Placement

Aim

The aim of this chapter is to review the design guidelines for placing gates and to run the gate location analysis for the cover model and other parts.

Why do it

Placing a gate correctly can be one of the most critical factors in determining the final quality of the part. The location of the gate may have many requirements and restrictions including part design, usage, aesthetics, and tool construction.

Overview

A review of design guidelines will be done to give a background for determining the gate location. The gate location analysis will be run on several different parts and the results will be compared to the requirements of the part. The gate location analysis can be done on Midplane and Fusion models. 3D flow simulation does not support this type of analysis.

Once a gate location is chosen, a fast flow analysis can be run to determine if the gate location is acceptable.

Theory and Concepts - Gate Placement

Guidelines for gate placement

There are many guidelines that are used to determine gating locations, depending on many factors including:

- Achieve a filling pattern that is:
  - Balanced.
  - Unidirectional.
- Place gates:
  - In thicker areas.
  - Far from thin areas
  - To prevent jetting.
  - To prevent weld line form forming in weak regions, or where they will be visible.
  - To prevent gas traps.
- Add gates to:
  - Lower the pressure to fill.
  - Prevent overpacking in a local area of the part.
- Gate placement depends on:
  - The tool type.
  - The runner type, cold or hot.
  - The gate type, Edge, submarine, etc.
  - Tooling restrictions.

The gate location has a significant impact on the filling of the part. Several examples of different gate locations are described below including:

- Gates on the end of the part.
- Gates in the center of the part.
- Two gated with a uniform flow length.
- Two gated with gates closer to the center of the part.
- Gates in thicker areas.
- Gates in thinner areas.
- Gates placed to achieve a balanced filling.
- Gates added to reduce pressure to fill.
- Gates added to balance the packing.

End-gated part

The end-gated part is considered balanced because the flow is unidirectional, and material continues to flow through every area of the part once it is filled (with the possible exception of the extreme left corners), shown in Figure 305. Placing a gate on the end of the part will produce an orientation that is aligned down the axis of the part. This type of gate location generally reduces warpage, in particular with amorphous and fiber filled materials. The disadvantage is that the flow length is quite long, so fill pressures will be relatively high, and packing may be a problem. With constant packing pressures, the variation in volumetric shrinkage can be high but this problem can be overcome with a decaying packing profile.

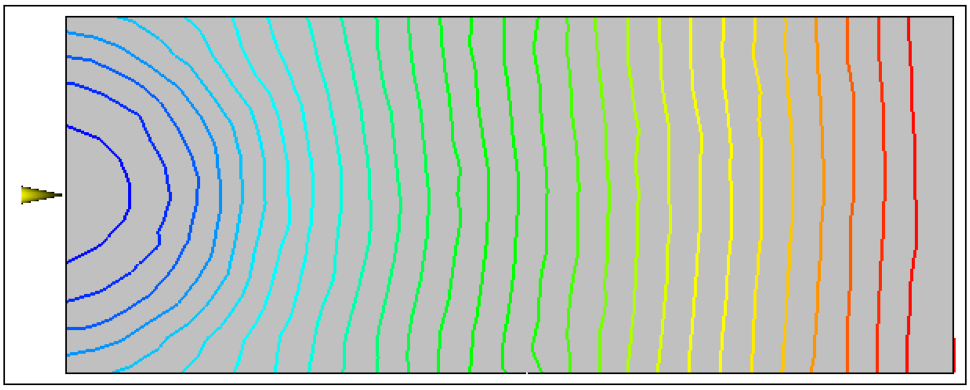


Figure 305: End gate with balanced filling

Center-gated part

The center-gated part in Figure 306 below is reasonably well-balanced but not as good as the end-gated part. The problem is that the flow front starts out to be radial but then straightens out and becomes linear. There is some degree of underflow because the flow length to the middle of the long side is very short compared to the flow length to the long end of the part. This can result in warpage, depending on the material and structure of the part. A center-gate location is better for round or square parts.

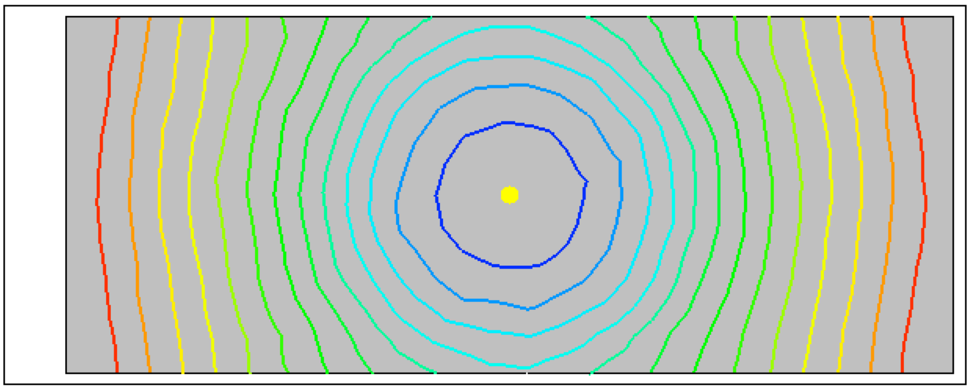


Figure 306: Center gate, mostly balanced

Two gates – uniform flow length

If a second gate is needed on the part, then the positioning of the two gates is critical. In Figure 307 below, the gates are placed so the flow length between the gate and end of the part is the same as the flow length to the weld line. The spacing was calculated by breaking up the length into twice as many sub-moldings as the number of gates. The gates are placed at the boundary between every other sub-molding. This gives the best possible balance within the part with multiple gates. Whenever gates are added to the part, each gate should fill about the same flow length and volume. This is difficult, and sometimes impossible with non-symmetrical parts, but this should be the initial goal.

One potential problem with this gate location is the weld line. The temperature of the flow front and the pressure on the weld line when it forms determines the quality of the weld line. With this gate location, the weld line forms at the end of fill, therefore the pressure drop between the gate and weld line will be higher than if the gates were closer, and potentially at a lower temperature when the weld line forms.

When considering warpage, this gate configuration will not over-pack the center of the part, possibly reducing the warpage of the part.

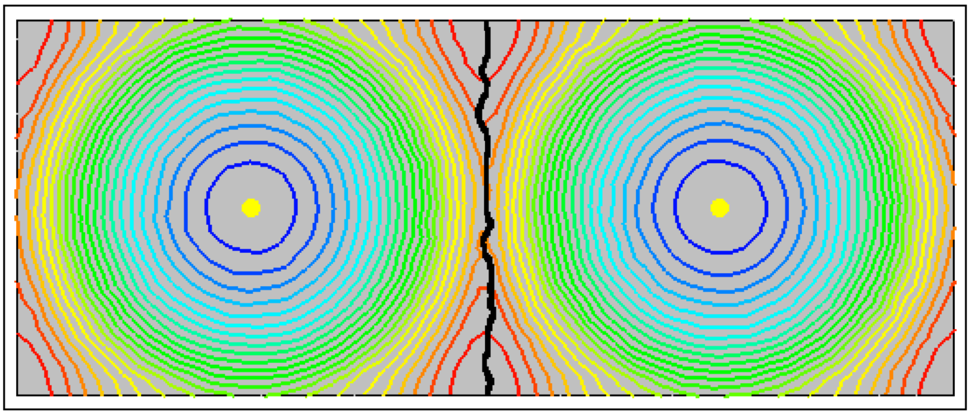


Figure 307: Two gates with uniform flow length, mostly balanced

Two gates – closer to the center of the part

This gate configuration shown in Figure 308, is very similar to the previous one. The gate spacing is calculated by splitting the length into a number of sub-moldings equal to the number of gates plus one. This places the gates closer together and the flow length to the ends of the part is longer.

As a result, there is overpacking between the gates possibly leading to warpage. The potential for warpage makes this gate configuration less desirable than the one in Figure 307; however, the weld line may be of higher quality than the previous case. This is because the weld line is formed closer to the gate at higher temperatures, and will see higher pressures. If quality of the weld line is of primary importance, this gate location may be better than the previous example.

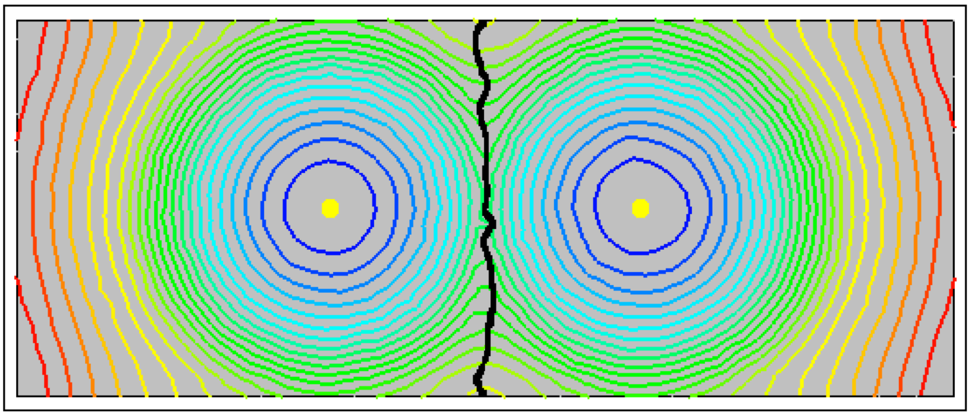


Figure 308: Two gates closer to center, not balanced

Gate in thicker areas

In Figure 309 below, the part has a 5 mm thick section and a 2 mm thick section. An edge gate was placed in the thin section on the part to the left, and in the thick section on the part to the right. Each part has the same gate, runner size, and processing conditions. The results indicate that the part with a gate placed in the thick section has much lower and uniform volumetric shrinkage compared to the part with a gate placed in the thin section.

Depending on the objectives of the part, it may be beneficial to place a gate in a thicker area even if the balance of the part may not be quite as good. This situation will most likely be the case if the material is semi-crystalline and/or if sink marks and voids are critical defects to be avoided.

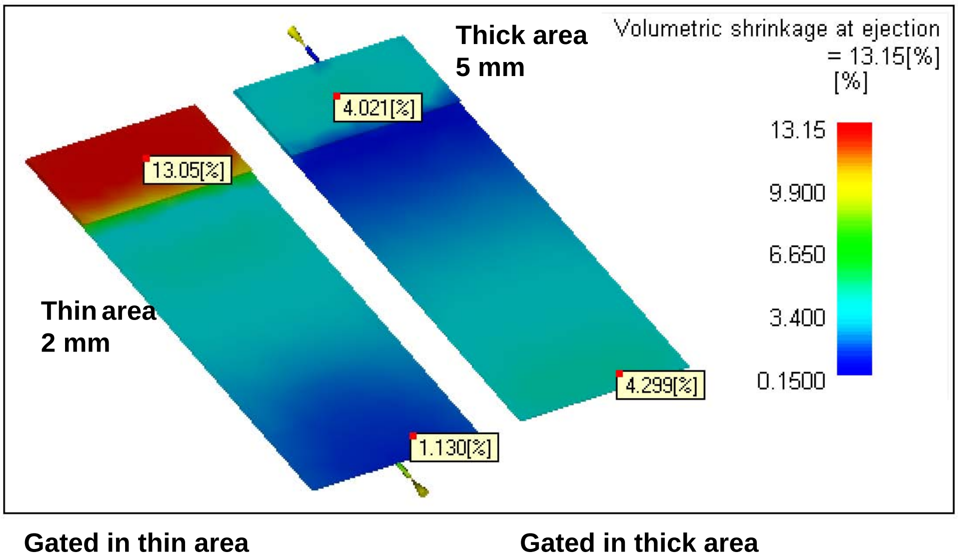


Figure 309: Gating in thin and thick sections, comparing volumetric shrinkage

Gate far from thin features

When there is a wide variation in wall thickness, place the gate as far as possible from thin features to avoid hesitation. In Figure 310 and Figure 311 below, the nominal wall is 2 mm and the rib is 1 mm thick. Both have the same processing conditions.

In Figure 310, the gate is close to the thin rib. When the flow front reaches the rib, the flow splits because the polymer takes the path of least resistance. Since the pressure required to go into the thick nominal wall is much less than the thin rib, most of the material flows in the nominal wall. The material flowing into the rib is hesitating, so there is little shear heat, and the material gets quite cold and eventually freezes off creating a short shot.

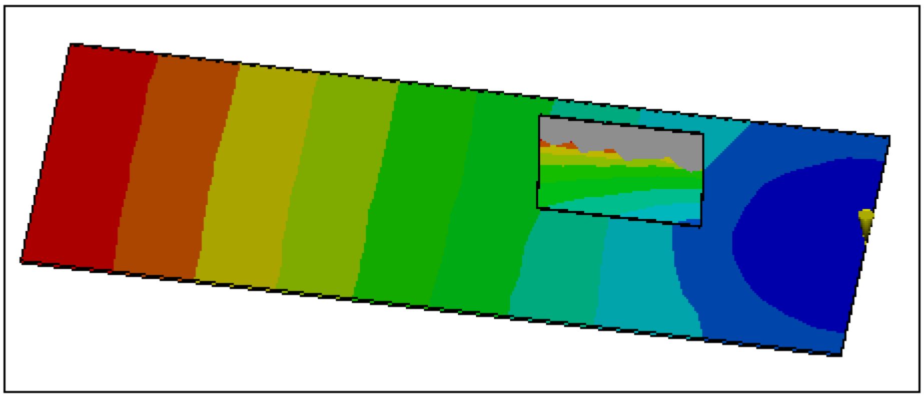


Figure 310: Gate close to thin feature

In Figure 311, the gate is at the far end of the part. In this case, when the material gets to the thin rib there is not much of the part left to fill. The material still hesitates going up the rib but there is not enough time for the rib to freeze off. The last place to fill is in the rib, but it does fill. This problem is more likely to occur with semi-crystalline materials, as they tend to freeze faster.

You can avoid or minimize problems with hesitation by using fast injection speeds and by reducing the thickness variation within the part.

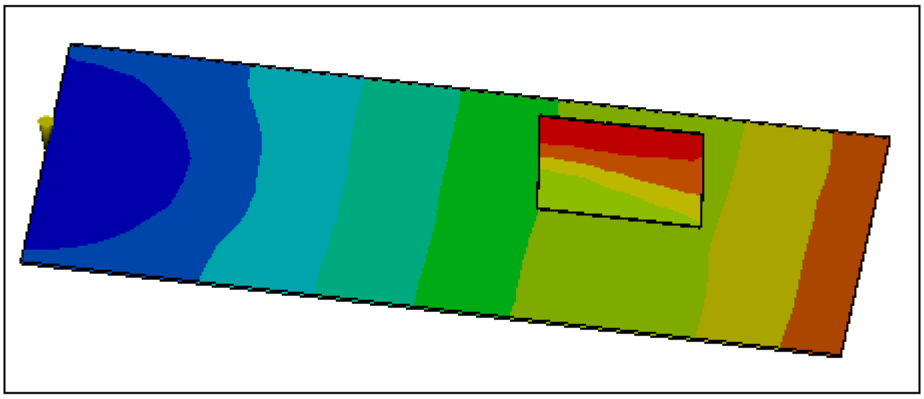


Figure 311: Gate far from thin feature

Place gates to achieve unidirectional filling

Unidirectional filling of the part is recommended, as shown in Figure 312. When the flow front does not change directions during the filling, there is little or no underflow. The molecular orientation and fiber orientation will be more consistent as well, which can make a huge difference in the warpage of the part. This is most critical in amorphous and glass fiber filled materials. Because unidirectional filling normally means placing a gate on the end of the part, it has the same possible disadvantages as the end gate discussed earlier.

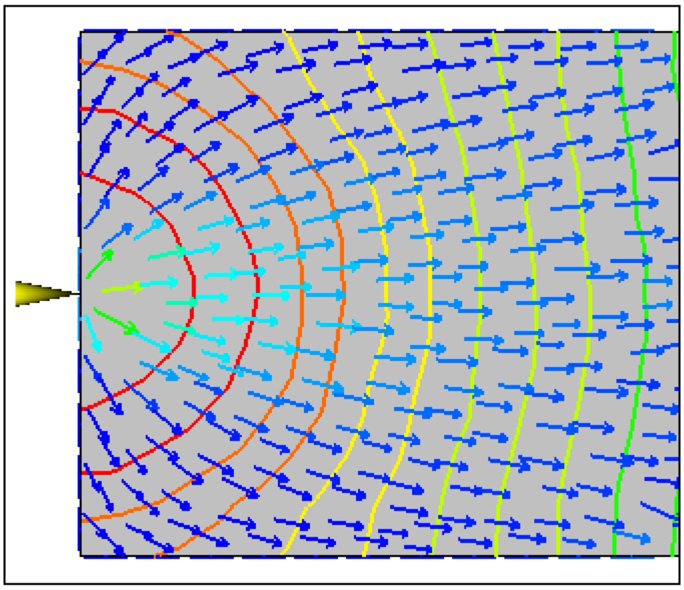


Figure 312: Unidirectional filling

Add gates as necessary to reduce pressure

Gates often need to be added so that the fill pressure is within the capacity of the injection molding machine. As a general rule, the fill pressure of the part without a feed system should be about **half the machine limit**, which is about **70 MPa (10,000 psi)** for a typical machine. When a single gate has a pressure that exceeds a pressure limit or guideline, you must lower the pressure. Changing the gate location to reduce the maximum flow length in the part is a good way to lower the pressure to fill. Once you have achieved the shortest possible or practical flow length and the pressure is still too high, add a second gate.

When adding gates, place them so all gates fill about the same volume and have about the same flow length. This will reduce the pressure. Figure 313 has one gate in the center of the part so it has the shortest possible flow length with one gate. The pressure to fill is above the capacity of the molding machine, so it is not possible to use the one gate.

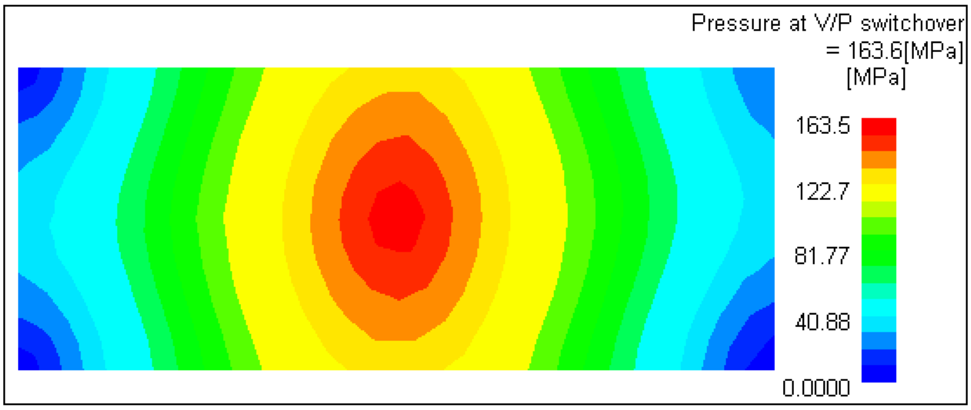


Figure 313: One gate - pressure too high

In Figure 314, a second gate has been added. The locations of the gates were determined so the flow length for each gate is the same to the edge of the part and to the weld line.

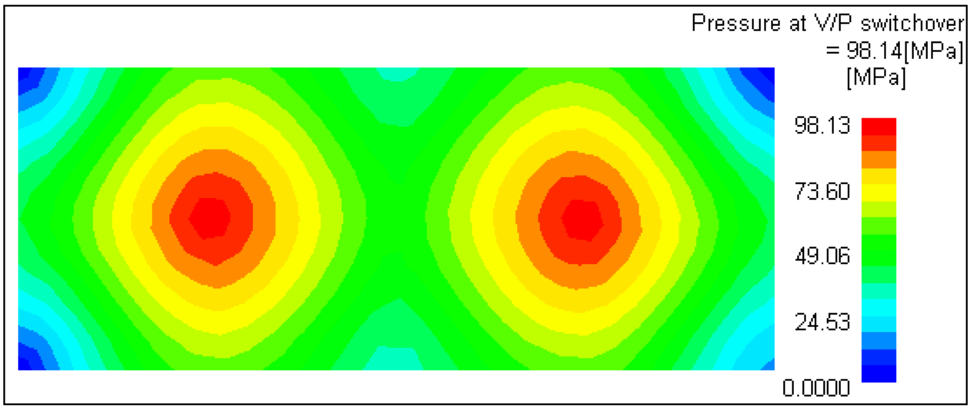


Figure 314: Adding a gate to reduce pressure

Prevent overpacking by adding gates

Depending on the geometry of the part, sometimes adding gates can improve the packing of the part by improving its uniformity. In Figure 315, the single-gated part has a balanced filling pattern. However, due to the center rib close to the gate, the volumetric shrinkage in the rib is very low because it is overpacked. This may cause a problem with warpage, but it also may cause a problem with ejecting the part. There may be other overriding factors in the decision to place a gate, but overpacking may be important.

Figure 316 adds a gate so the center rib fills near the end of fill. The volumetric shrinkage in that center rib is much better than the single gate location.

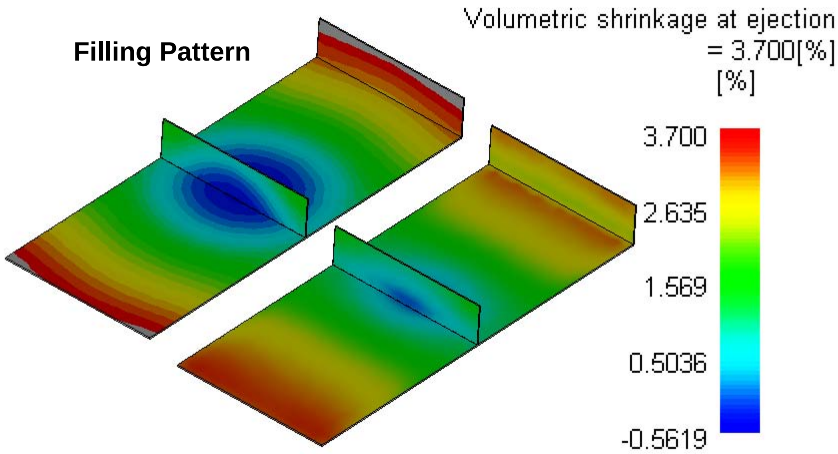


Figure 315: Center gate, overpacked center rib

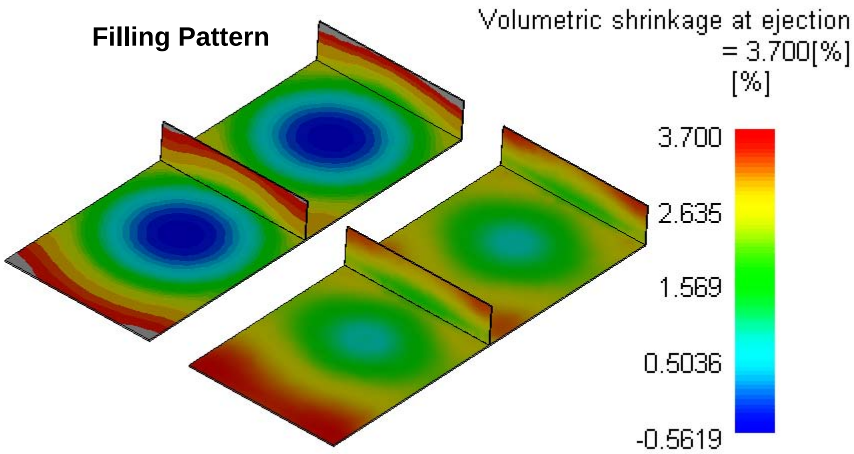


Figure 316: Two gates reduce overpacking in center rib

The type of tool being used

For a given part design, several different types of tools could be used to make the part, including:

- Is it a 2 plate or 3 plate?
- Are the runners cold, hot, or a combination?
- What type of gates are desired – edge, tunnel, etc.?
- What restrictions are there for where the gate can or cannot go due to the part design?
- Does the cavity layout restrict gate locations?
- Are there tooling features that restrict gate locations, such as slides or lifters?

There may be many things that limit where you can place the gate besides strictly flow- or warp-related concerns. Using flow simulation, you can find the best gate location within the constraints of the part and tool design.

Gate location analysis overview

The **Gate Location** analysis sequence will calculate a possible gate location based on the following factors:

Processability

Is it possible to produce a part if the part is gated at this location? This is the major component in the Gate Locator analysis. If the part cannot be produced from an investigated location, the location will appear red. Otherwise all other components will be factored into the result.

Minimum Pressure

Lower injection pressure usually produces lower shear rate and shear stress levels, or lower clamp tonnage requirements.

Geometric Resistance

Where would a gate not cause overpacking? For multiple gated parts, this calculation measures the resistance through each node. If two gates fill with equal flow resistance, then they fill with equal pressure producing no overpacking. If the resistance is different then one gate will fill first resulting in an overpacking situation whilst the other gate continues to fill.

Thickness

Is it possible to pack the part effectively when gating at this location?

Running a gate location analysis

The Gate Location analysis sequence can be set by clicking the Analysis Sequence icon , or by the menu command **Analysis** ➡ **Set Analysis Sequence.** Analysis inputs fall into two categories; mandatory and optional. Reviewing the optional inputs will give you a better understanding of how the analysis works.

Mandatory inputs for a gate location analysis

A gate location analysis will be one of the first analyses you run. As a result, there are not many inputs required. Inputs that are required include:

- Meshed study model.
- Analysis sequence of **Gate location.**
- Material.

These are the only parameters that must be entered to run an analysis. Normally these are the only inputs used.

Optional inputs

The optional inputs except Injection location are set from the Process Settings Wizard, shown in Figure 317.

Injection location(s)

The analysis can work with any number of injection locations, including zero. Normally a gate location analysis occurs without an injection location. When the analysis has no defined injection location, the analysis will indicate the location for the best single gate. When one or more injection locations are defined, it will indicate the location for the next gate. This is called the one + one technique.

Molding machine

The default injection molding machine is normally used. If desired, you can choose the specific machine from the molding machine list, or you can change a molding machine definition. The important parameters are the pressure and clamp force limits. The pressure and clamp tonnage limit are used in the gate location analysis. If the pressure is too high, that area will not be considered for a gate location.

Mold surface temperature

This is the same variable used for a normal fill analysis. It is the plastic/metal interface temperature.

Melt temperature

This is the temperature of the polymer entering the part.

Advanced option – Injection pressure limit

Automatic or specified, in advanced options. The automatic limit is 80% of the value in the machine database for the selected machine. A specified value overrides the automatic value.

Advanced option - Clamp tonnage limit

Automatic or specified, in advanced options. The automatic limit is 80% of the value in the machine database for the selected machine. A specified value overrides the automatic value.

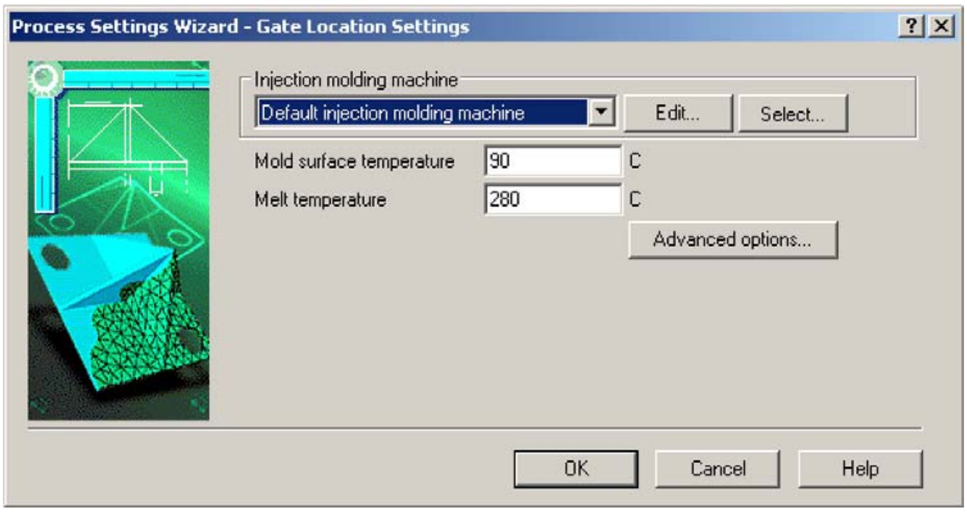


Figure 317: Gate location process settings

Gate location analysis results

The results available include:

- Screen output.
- Results summary.
- Best gate location plot.

Screen output and results summary

The screen output and results summary logs for the gate location analysis are essentially the same thing. Both contain a node number representing the best gate location. If there is no suitable gate location due to a pressure or clamp tonnage limit, the Screen output indicates this, and the best gate location plot shows only red (which indicates that there is no ideal gate location).

```
Copyright Moldflow Corporation and its worldwide subsidiaries. All
rights reserved.
(C)2000 2001 2002 2003 2004 2005
This product may be covered by
   US patent 6,096,088 ,
   Australian Patent No. 721978 ,
and foreign patents and pending applications

Gate Location Analysis

Version: mpi600  (Build 05512)

Analysis running on host: MF-JAY
        Operating System: Windows XP Service Pack 2
          Processor type: GenuineIntel x86 Family 15 Model 2 Stepping
9~3192 MHz
    Number of Processors: 2
   Total Physical Memory: 1022 MBytes

Analysis commenced at        Sun Jan 15 10:15:14 2006
Flow is using stored mesh match and thickness data
Match data was computed using the maximal-sphere algorithm

   Maximum design clamp force                        =     5600.18
tonne
   Maximum design injection pressure               =      144.00 MPa
   Recommended gate location(s) are:
      Near node                                      =        1845
Execution time
   Analysis commenced at        Sun Jan 15 10:15:14 2006
   Analysis completed at        Sun Jan 15 10:16:04 2006
   CPU time used                       48.17 s
```

Figure 318: Sample Screen Output log

Best gate location plot

The best gate location plot uses a scale from 0.1 to 1, with 0.1 corresponding to the worst gate location (red), and 1 corresponding with the best gate location for the part (blue).

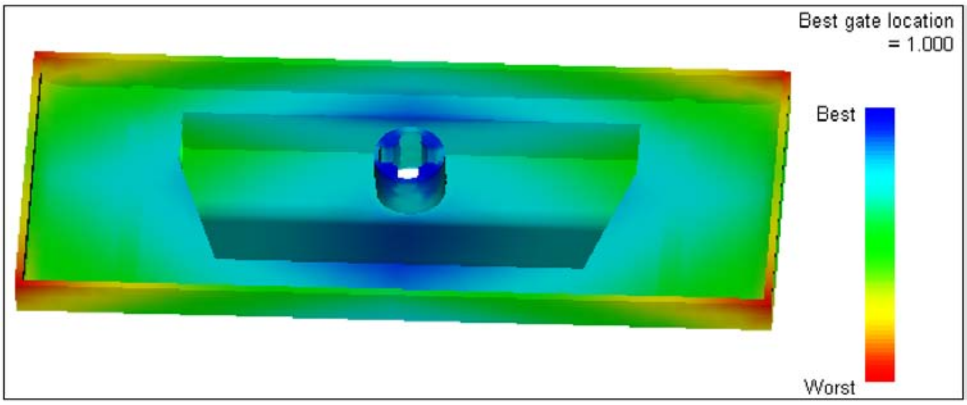


Figure 319: Cover gate location results

Results interpretation

The gate location analysis does not take into account some practical considerations for the gate location such as the type of tool, or limitations on the gate location due to part aesthetics etc. In Figure 319, this part is limited to a 2-plate tool with a cold runner system. As a result of that tooling restriction, the suggested best gate location for the cover, in the middle of the part, is not possible. These results can only be used as a guide to place a gate on the parting line.

Using a best gate location

If the best gate location is a location you would like to use for a flow analysis, follow the steps listed below.

1. Click on **Logs** box in the Study Tasks pane.
2. Click on the **Screen Output tab**.
   2.1. Look near the bottom of the file.
   2.2. Find the recommended gate location, which will be listed as a node number as shown in Figure 320.
3. Click **Modeling ➨ Query Entities**.
   3.1. Enter the node number of the recommended gate location in the entity identify field.
   3.2. Place the letter **N** in front of the number; for example, **N1845**.
   3.3. Check the box for **Place results in diagnostics layer**.
   3.4. Click **Show**.
   - The node is placed in a **Queried entities** layer. Show and hide the layers as needed to see the exact location of the node. Initially, the node may be difficult to see but by hiding layers the queried node will be found. If this location is usable, set the injection location on this node.
4. With the best gate location node on the queried entities layer, click the injection location icon  and select the node.

5. When prompted, click **Create copy** to prevent deleting the gate location results.
    - This occurs because the injection location command was executed and an injection location can't be selected for a study with results.

```
Recommended gate location(s) are:
      Near node                                       =        1845
```

Figure 320: Example of the screen output for a gate location analysis

Gate location validation

Once you have chosen a gate location, consider running a **Fast Fill** analysis to validate the gate location. The Fast Fill analysis is better to use than the standard Fill analysis because of its speed. The fast analysis will allow you to look at the filling pattern and other key results. If your original gate location is not optimum, then you can modify the gate location and run the analysis again. Once you are happy with the gate location, a run a standard Fill analysis to verify the gate location further.

The Fast Fill analysis is a non-isothermal, non-Newtonian, incompressible analysis. It has a reduced number of laminates with only a small group of results available, including:

- Fill time.
- Pressure at V/P switchover.
- Temperature at flow front.
- Time to freeze.
- Air traps.
- Pressure at end of fill.
- Weld lines.

The analysis is available for both Midplane and Fusion models. To specify the analysis type, select **Analysis** ➡ **Set Analysis Sequence** ➡ **Fast Fill**. You can also set this in the Study tasks pane.

The **Process Settings Wizard** for a Fast Fill analysis shown in Figure 321 is slightly different from a typical Fill analysis. On this wizard page there are options for the machine clamp force and injection pressure limits, and there are no advanced options to use.

Use the Fast Fill analysis for the following reasons:

- As a quick quality check of the model.
- To optimize the gating locations.
- As an initial check of process settings.
- To test valve gate timings.

The Fast Fill analysis is not meant to replace the multi-laminate fill analysis.

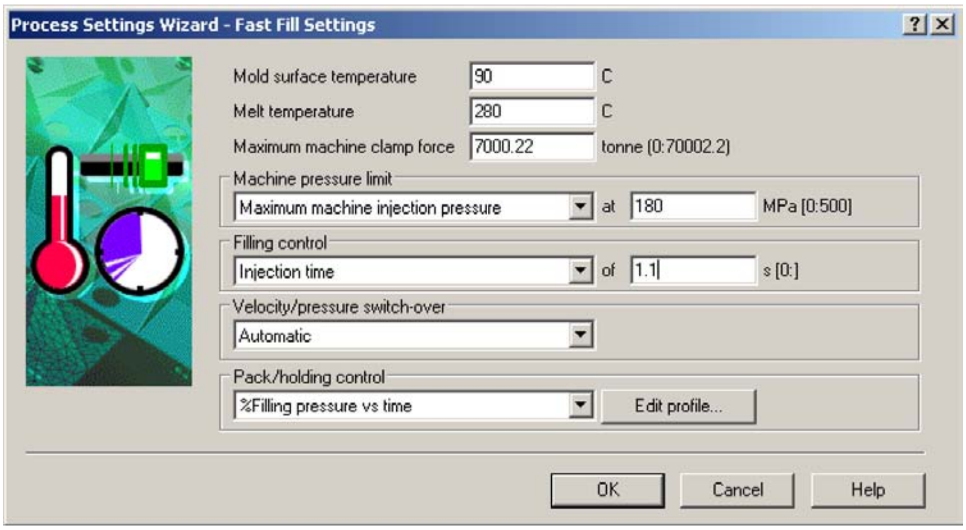


Figure 321: Fast Fill analysis process settings

Gate location analysis on 3D models

The gate location analysis is not available for 3D models. However, a Fusion version of the model can be created for use with gate location analysis. The match ratio may not be very high, but for the preliminary work of the gate location analysis and even a Fast Fill analysis it should be good enough. The knowledge gained from the gate location analysis can be applied to the 3D model.

What you've learned

There are a number of guidelines used to help determine the gate location including:

- Gates on the end of the part.
- Gates in the center of the part.
- Two gated with a uniform flow length.
- Two gated with gates closer to the center of the part.
- Gates in thicker areas.
- Gates in thinner areas.
- Gates placed to achieve a balanced filling.
- Gates added to reduce pressure to fill.
- Gates added to balance the packing.

However, practical considerations often prevent the gate from being in the best gate location from a flow perspective. The best gate location should be used with respect to flow considerations within other limitations.

The gate location analysis does not take into consideration all of the gate location guidelines nor any other restrictions. Often, its recommendations are not possible, but may offer some guidance with the gate placement.

The fast fill analysis is an excellent tool to review gate locations, no matter how the are determined. It runs very quickly and is best used as an initial check at the overall filling pattern for the part.

CHAPTER 14

Molding Window Analysis

Aim

The aim of this chapter is to find the optimum molding conditions and widest possible molding window for the parts provided.

Why do it?

The optimum molding conditions for the part, i.e. mold temperature, melt temperature and injection time, is a good starting point of a finite element flow analysis. A molding window helps determine how much variation you can have in mold and melt temperatures, and the injection time and still make a good part. In addition, several gating locations can be evaluated with a molding window analysis. You can determine if a particular gating location is possible, or which one gives a wider molding window.

Overview

Molding conditions are evaluated on a part by running a quick analysis on the part. The analysis is done with a simple model that represents the thickness, flow length and volume of the part. This model is automatically built on the midplane or Fusion model. To run this analysis, a gate location and material must be defined. This simple model can be analyzed at a small fraction of the time of the midplane or fusion model, saving a significant amount of time.

The output of this quick analysis includes an “optimum” set of conditions, plots indicating the size of the molding window and plots showing how part quality, injection pressure, shear stress, flow front temperature, cooling time and shear rate are influenced by changes in processing conditions.

Results from the analysis can be reviewed to determine if the optimum conditions should be used, or if there should be changes in the molding conditions. If several different gate locations are to be compared, results from the different gate locations need to be evaluated to determine which gate location is best.

CHAPTER 15

Fiber Flow Analysis

Aim

The aim of this chapter is to be introduced to the MPI/Fiber analysis program. This includes background theory about fibers, why the fiber-orientation analysis is important, and how to run an analysis and interpret the results.

Why do it

A fiber flow analysis is a standard flow analysis plus an algorithm that calculates the distribution of short fibers in a polymer matrix. For conducting a warpage or stress analysis of a filled material, it is critical that the distribution of the fibers be considered as they dominate the physical properties of the material.

Overview

Running a fiber flow analysis is a simple task. If the material is a fiber filled material, the database contains all of the additional data necessary to run this material. By default, a fiber flow analysis is run unless you indicate that the fiber-orientation flow analysis should not be run. In this chapter, you will review fiber flow analysis results on a Fusion or Midplane part.

Theory and Concepts - Fiber Flow Analysis

What is MPI/Fiber?

MPI/Fiber is a separate flow analysis module that in addition to calculating the flow of a fiber filled material; it predicts the distribution of the fibers within the polymer composite. The resulting thermo-mechanical properties of the composite are calculated also. When running a warpage analysis, it is important to determine the distribution of fibers within the composite, as the fibers normally dominate the shrinkage and warpage of the composite.

To run MPI/Fiber, there must be an MPI/Flow and MPI/Fiber license available. Fiber can be used with 3D, Fusion and midplane model geometries. The main output from Fiber is not graphical results but interface files for MPI/Warp and MPI/Stress (midplane only). If an analysis is not going to continue to Warp or Stress, there is little point in running the fiber analysis.

Why Run Fiber?

Fiber fillers significantly change the performance of the polymer. Predicting the effect a fiber has on the polymer is important. The reasons MPI/Fiber should be run include:

- Fiber orientation is usually the main cause of warpage in a fiber filled material.
- Fibers change the mechanical properties of the polymer therefore their reaction under structural loads.
- Fiber orientation varies with flow direction, thickness and part geometry.
- Predicting the orientation of fiber orientation is important because the fiber orientation influences mechanical properties an anisotropic manner.
- Predicting the fiber orientation is critical to the part function.
- Having the ability to alter the part and/or mold design and then predict orientation is crucial.

Fillers and Fibers

Fillers change the polymer composite's properties improving in several possible ways including:

- Increased stiffness.
- Increased strength.
- Reduced creep and stress relaxation over time, i.e. time dependant properties.
- Increased upper temperature use limits.
- Improved dimensional stability.
- Possibly reduced material costs.

- Increased electrical and thermal conductivity.

Types of fillers

Reinforcing fibers

One of the most common types of fillers is fiber. There are many types of materials used in fiber including, glass, carbon, fibrous minerals, boron, Kevlar. Fibers add many useful properties to the polymer composite including:

- Increased flexural stiffness (modulus).
- Increased tensile strength.
- Reduced creep and stress relaxation over time.
- Increased heat deflection temperature.
- Improved dimensional stability (reduced warpage).

Conductive fillers

Conductive fillers are often used in electrical and shielding applications where static needs to be dissipated or electricity needs to be conducted. Common fillers include, carbon fiber, graphite and aluminum powers.

Filler vs. Fiber

All fibers used for injection molded parts are fillers, but not all fillers are fibers. The difference between a filler and a fiber is the aspect ratio of the filler. A fiber has length to diameter ratio of greater than one. Some fillers have aspect ratios less than one, or one. Only fillers (a fiber) that has an aspect ratio greater than one can be used with MPI/Fiber. Fibers that come in the form of flakes such as, glass, metal and mica, have an aspect ratio of less than one. Fillers in the form of cubes or spheres including talc and minerals, have an aspect ratio of one. Only fibers have an aspect ratio of more than one. Common fiber materials are glass, carbon and Kevlar. The typical aspect ratio of these materials is 25, however they can get as long as $5E^6$. MPI/Fiber is designed to work with short fiber-filled materials. If composite is made with long glass fibers, the processing of the polymer through the barrel, nozzle and feed system will break up the fibers making them shorter and appropriate to be used with MPI/Fiber.

Results from MPI/Fiber

The results of MPI/Fiber include all the standard flow results plus results related to the fiber orientation and it's affect on the composite. Results from MPI/Fiber are described below.

Average fiber orientation

This result shows the average fiber orientation through the thickness of the element over time.

Fiber orientation tensor

The fiber orientation tensor shows the distribution of fibers at different laminates through the thickness at the end of the cycle.

Poisson's ratio

Poisson's ratio defines strain in the second principal direction caused by the stress in the first principal direction. There are two versions of this result, one is an element average (fiber) and the other is the profile result over the thickness, or in other words, the value per laminate (v12).

Shear Modulus

This is the ratio of shear stress to shear strain. There are two versions of this result, one is an element average (fiber) and the other is the profile result over the thickness, or in other words, the value per laminate (G12).

Tensile Modulus, 1st and 2nd principle directions

The 2nd principle direction is perpendicular to the 1st principle direction. This is the ratio of stress to strain in tension. The 1st principle direction coincides with the fiber orientation first principal direction. There are two versions of this result, one is an element average (fiber) and the other is the profile result over the thickness, or in other words, the value per laminate (E1 and E2).

Linear Thermal Expansion Coefficient, 1st and 2nd principle directions

Linear Thermal Expansion Coefficient is the ratio of the change in length per degree K. The 1st principle direction coincides with the fiber orientation first principal direction. There are two versions of this result, one is an element average (fiber) and the other is the profile result over the thickness, or in other words, the value per laminate (a1 and a2).

Orthographic set

Please note that the above mentioned properties are part of the whole set of the thermo-mechanical properties based on orthotropic assumption. Since MPI/WARP uses the shell element model for midplane/fusion analysis, the "basic properties" mentioned above are enough for such a shell structural analysis. However, the whole set of properties can be produced if the fiber-filled property output is set to **orthotropic set**, on the **Composite Property Calculation Options** dialog as shown in Figure 348. This dialog is accessed by the Fiber Orientation Solver Parameters dialog. The whole set of properties includes all nine components (E1,E2,E3,v12,v23,v13,G12,G23,G13) of the mechanical properties and three thermal expansion coefficients (a1,a2,a3) for laminated results. In MPI/FIBER3D analysis, the whole set of the thermo-mechanical properties is always produced on elemental basis as MPI/WARP3D needs the whole set.

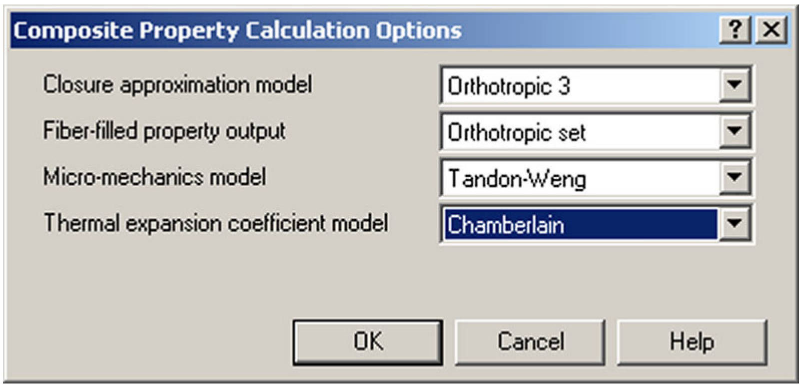


Figure 348: Composite property calculation options

Definition and prediction of fiber orientation

Calculation of three dimensional fiber orientation is performed concurrently with the mold filling analysis on the same finite element mesh. Each triangular element may be considered as consisting of several laminates subdividing the local molding thickness. Each laminate is identified by the grid point through which it passes. The midplane of the molding passes through grid point 1. An orientation solution is calculated at each laminate for each element in the mesh. In this way, it is possible to observe the variation in orientation distribution on a set of planes parallel to the mold surface through the cross-section of the molding.

The three-dimensional orientation solution for each element is described by a second order tensor. The fiber orientation tensor is derived from the concept of fiber angle probability distribution, as shown in Figure 349, and the orientation average, of a single fiber unit vector's component products. For graphical representation, the fiber orientation tensor a_{ij} in any coordinate system can be described with its nine components in the coordinate system (tensor A of Figure 350), or by its eigenvalues (λ_1, λ_2, and λ_3 (Tensor B of Figure 350) in a coordinate system described by its eigenvectors e1, e2, e3 (Tensor C in Figure 350). The eigenvectors indicate the principal directions of fiber orientation and the eigenvalues give the statistical proportions (0 to 1) of fibers aligned with respect to those directions. This information is used to define an orientation ellipsoid which fully describes the probability distribution of fibers for any location and / or time of interest. A general orientation ellipsoid is also shown in Figure 350.

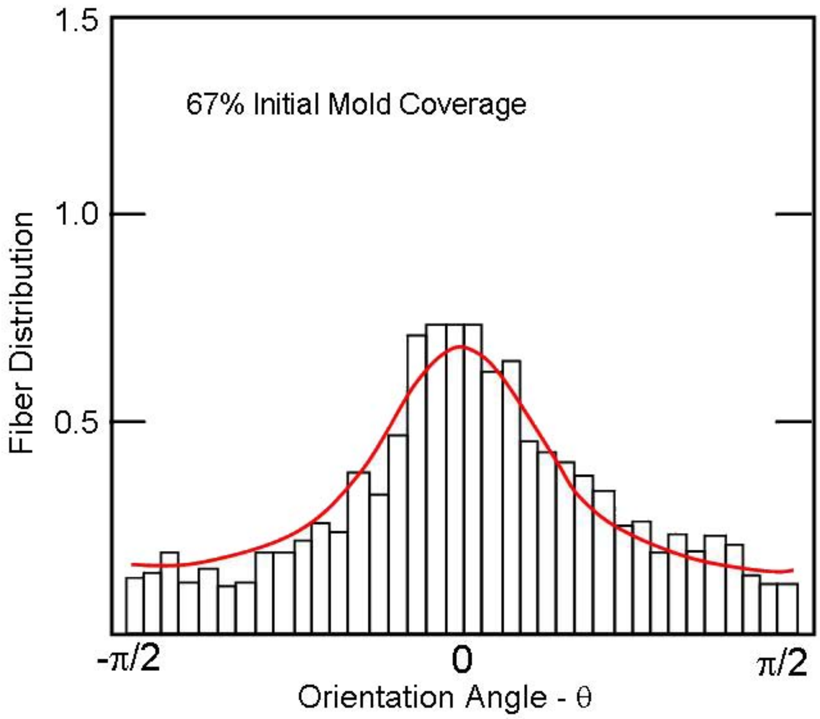


Figure 349: Schematic view of fiber angle distribution [1]

$$a_{ij} = \underset{\textbf{Tensor A}}{\begin{bmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{bmatrix}} \rightarrow \underset{\textbf{Tensor B}}{\begin{bmatrix} \lambda_1 & 0 & 0 \\ 0 & \lambda_2 & 0 \\ 0 & 0 & \lambda_3 \end{bmatrix}} ; \underset{\textbf{Tensor C}}{\begin{bmatrix} e_1 & e_2 & e_3 \end{bmatrix}}$$

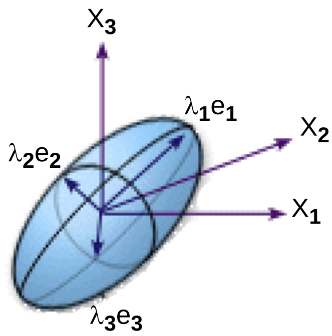


Figure 350: General orientation ellipsoid

For display purposes, this 3D ellipsoid is projected onto the plane of each element to produce a plane ellipse. This creates a useful representation of orientation distribution, since the gap wise orientation components eliminated by projection are usually small. In this representation, a near random distribution is displayed as an ellipse tending to a circle while, for a highly aligned distribution, the ellipse degenerates to nearly a line.

Description of the orientation tensor

The second order orientation tensor, a_{ij}, provides fiber orientation at any point of location and time. The tensor has nine components in three dimensional space, with the suffixes for the tensor components as already described in above section, and because its symmetric nature and the representation of probability distribution, only five components of the tensor are independent. The symmetric nature means:

$$a_{ij} = a_{ji}$$

which is the reason we only show the upper triangle components of the tensor in post-processing plot property dialog box as shown below, and here Figure 351.

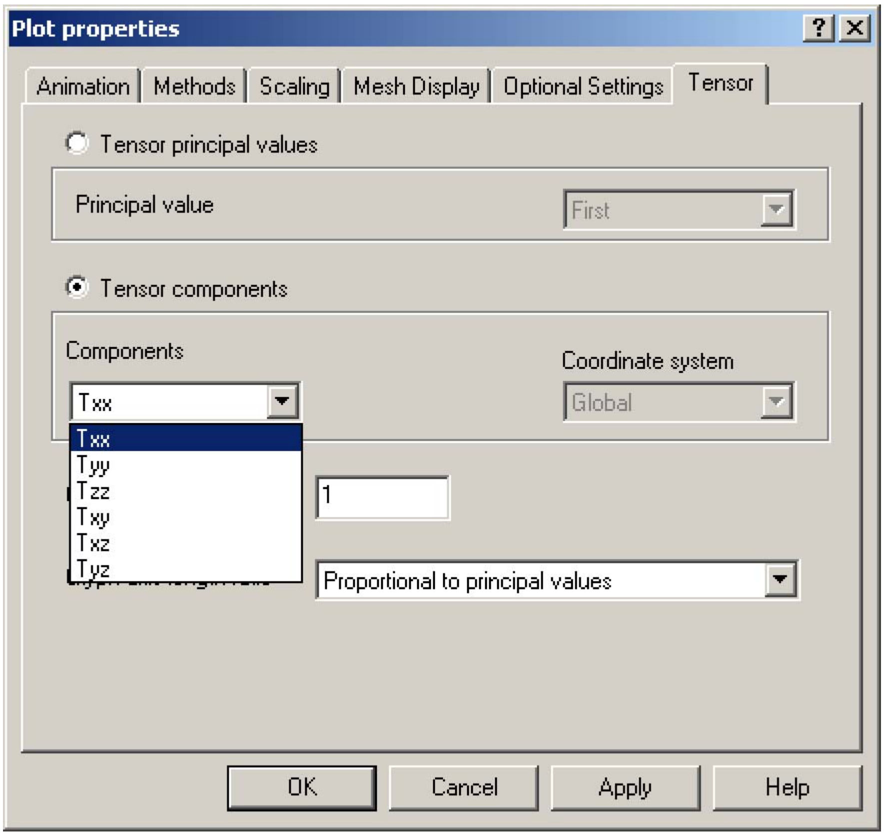


Figure 351: Tensor component selection option for display

The notation described above is different to the notation used in this plot property for the fiber orientation tensor and average fiber orientation results because tensor plot is generic and it can be used in setting up other tensor plot such as stress tensor. The table below summarizes the notation differences.

Table 51: Notation Conversion

Orientation Ellipsoid Notation	Fiber Plot Properties Notation
a_{11}	T_{xx}
a_{22}	T_{yy}
a_{33}	T_{zz}
a_{12}	T_{xy}
a_{13}	T_{xz}
a_{23}	T_{yz}

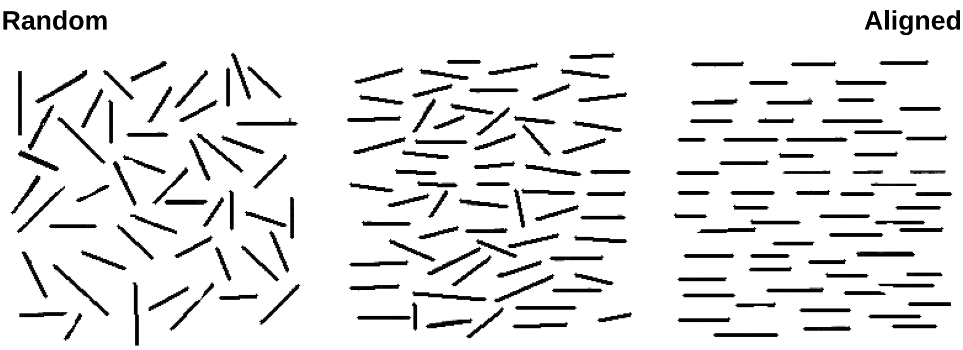
The probability distribution integrated in all directions has to be unity which means the normalization condition is $a_{11} + a_{22} + a_{33} = 1$

For planar orientation of the tensor, the out-of-plane components are gone so it has four components but only two are independent due to the two natures mentioned above.

Interpreting fiber results is discussed starting on page 534.

Using the fiber orientation results

Interpretation of fiber orientation result requires basic understanding of above mentioned orientation tensor concept. Figure 352 shows some example planar orientation states and the corresponding orientation tensors.

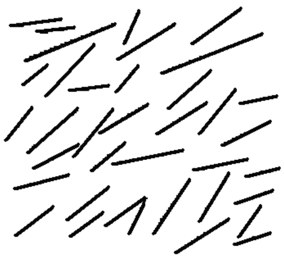


$$a_{ij} = \begin{bmatrix} 0.5 & 0.0 \\ 0.0 & 0.5 \end{bmatrix} \qquad a_{ij} = \begin{bmatrix} 0.7 & 0.0 \\ 0.0 & 0.3 \end{bmatrix} \qquad a_{ij} = \begin{bmatrix} 1 & 0 \\ 0 & 0 \end{bmatrix}$$

Figure 352: Planar fiber orientation states and the corresponding tensors [2]

Because the reference coordinate system used here implies that X-direction is the horizontal direction and if all fibers align in this X-direction means the probability distribution in this direction is 100% so the value a_{11} is 1 and all the other components are zero in the right-most situation. All these three situations show zero off-diagonal components, which means that the X direction coincides with the first principal direction and only the eigenvalues λ_1 and / or λ_2 are shown here non-zero.

If the principal direction is other than the x direction, as shown in Figure 353, then the off-diagonal component, such as a a_{12} in planar cases here, will be non-zero.



$$a_{ij} = \begin{bmatrix} 0.7 & 0.4 \\ 0.4 & 0.3 \end{bmatrix}$$

Figure 353: Fiber orientation represented by non-zero off-diagonal component

Fiber input to other analyses

Although the results are graphical, the main reason MPI/Fiber is run is to provide input into other analyses. Fiber results from midplane and Fusion and 3D are used for MPI/Warp. Midplane fiber results can also be used in MPI/Stress. The data that is used includes:

- Fiber orientation.
- Elastic moduli.
- Shear moduli.
- Poisson's ratios.
- Thermal expansion coefficients.
- In-cavity residual stresses.

These results can now be exported to other structural packages such as ABAQUS, ANSYS and LS-DYNA with the use of interface files.

Material property requirements

Fiber data

The following information is required for the filler/fiber so it can be used in MPI/Fiber:

• Filler type.	• Thermal conductivity.
• Weight fraction*.	• Elastic moduli.
• Aspect ratio.	• Poisson's ratios.
• Density.	• Coefficients of thermal expansion.
• Specific heat.	• Shear moduli.

*Volume fraction is converted in the code from weight fraction using filler/fiber and composite densities.

The filler/fiber properties of any material can be viewed from the **Materials Details** dialog. Click on the Fiber tab to see the properties of the material, click on the filler then hit the details button.

Polymer data

Mechanical data is required for the polymer itself, and may not be available. If it is available, it will be used. To edit or look at mechanical data for the polymer, go to: **Process Settings Wizard ➡ Advanced Options ➡ Molding material Edit ➡ Mechanical Properties tab ➡ Use matrix properties check box ➡ Edit matrix properties.** The matrix properties are the mechanical properties for the polymer. If the matrix properties are not available, the flow solver will calculate them.

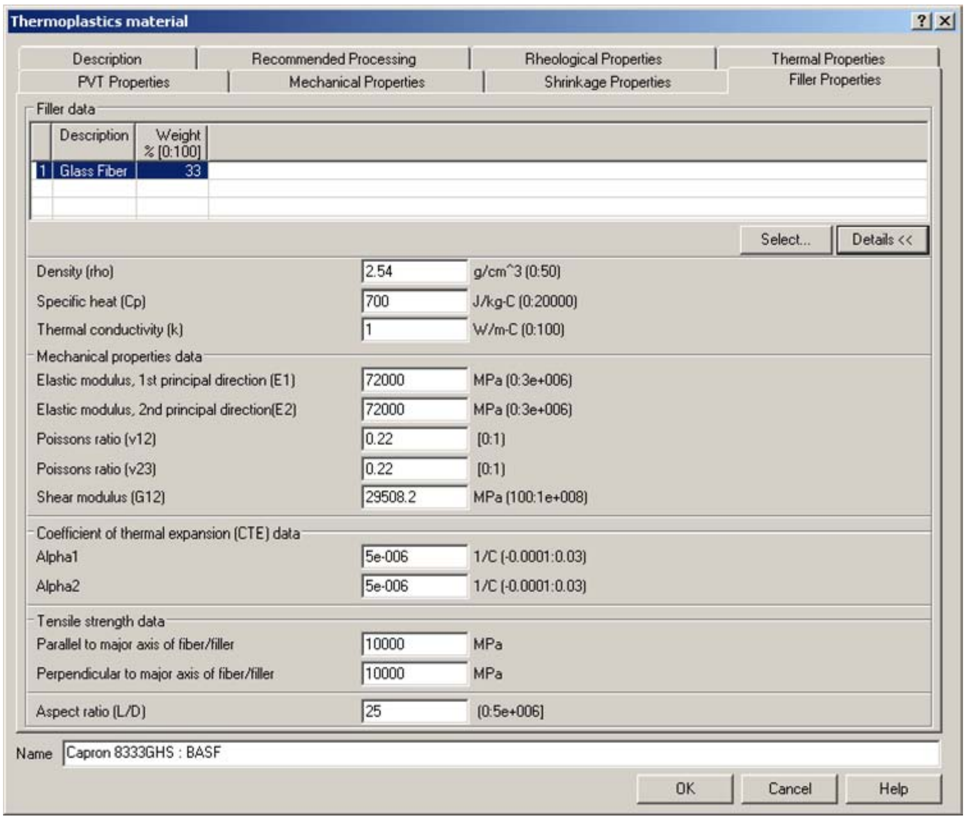


Figure 354: Fiber data properties dialog

Assumptions in MPI/Fiber

As mentioned previously, there is an assumption that the fiber is short. For long glass products, the assumption is that by the time the material goes through the barrel, nozzle and feed system the aspect ratio is short enough to be used. The aspect ratio also has to be greater than one. The solver also assumes the fibers are evenly distributed throughout the volume of the polymer.

Fiber and warp

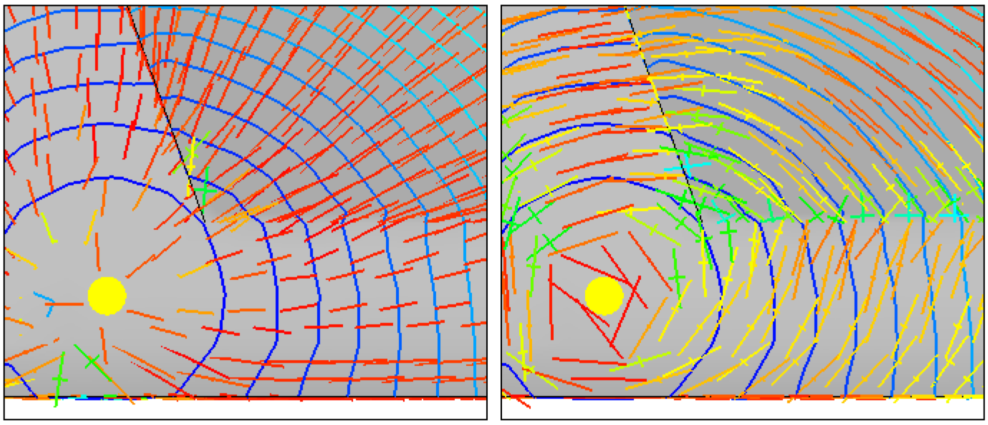
When a flow analysis with fiber is used as input to warpage, the fiber results are automatically used. The warpage results will be even better when shrinkage testing is done on the material. Warpage predictions are not reliable when MPI/Fiber is not used for a fiber-filled materials as the flow solver. A regular flow analysis without fiber analysis for the fiber-filled material even with CRIMS data cannot pick up the effect of the fiber orientation. To solve warpage problems, changing the gate location normally has the biggest impact on the warpage. Injection speed can also have an effect.

Fiber orientation

The orientation of fibers within a polymer is very complex. However, there are a couple of trends that take place that can help you understand how fibers are oriented. With the first trend, the fibers are oriented in the direction of flow in a high shear zone. In the portion of the cross-section where there is significant shear, fibers are oriented in the direction of that shear. This is the flow direction. This is shown in Figure 355 in the left plot.

With the second trend stretching flows tend to align the fibers in the direction of the stretching. This will occur in radial flow fronts. Once a fiber gets to the flow front of radial flow pattern, the forces are stronger in the radial direction so the fibers will be stretched radically or perpendicular to flow. This effect is dominant in the center of the cross-section. An example of this is in the right plot of Figure 355. As the fibers get closer to the mold wall, the shear will increase. At some point in the cross section, the shear forces will dominate over the stretching forces, so the fibers will be oriented more in the direction of flow. With radial filling patterns, fiber orientations can go from highly aligned in the direction of flow to random to highly aligned perpendicular to flow.

> On the Flow page of the Process Settings Wizard, there is a check box called **Fiber orientation analysis if fiber material** which is on by default. Users can opt to turn it off if only the flow analysis is interested in order to achieve faster turn-around solution.



Orientation due to a high shear rate **Orientation due to stretching flow**

Figure 355: Fiber orientation at different locations within the cross section

Another influence on fiber orientation is the geometry that the polymer flows through as seen in Figure 356. In section A, the fibers are entering and are randomized. As the flow front gets to a restriction, the fibers align in the restriction. As they exit, the stretching takes over and the fibers are aligned across the flow direction.

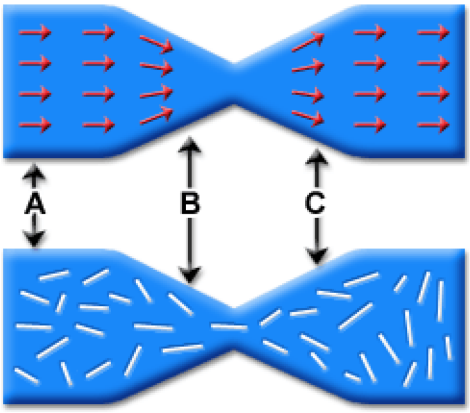


Figure 356: Converging and diverging flow

Using a fiber flow analysis

Running the analysis

When the licenses are available, running a fiber flow analysis is easy. By default it is run. On the Process Setting Wizard, Figure 357, there is a check box called, **Fiber orientation analysis if fiber material**. This is checked by default, so a fiber analysis will be run when the material if fiber filled, and licenses are available.

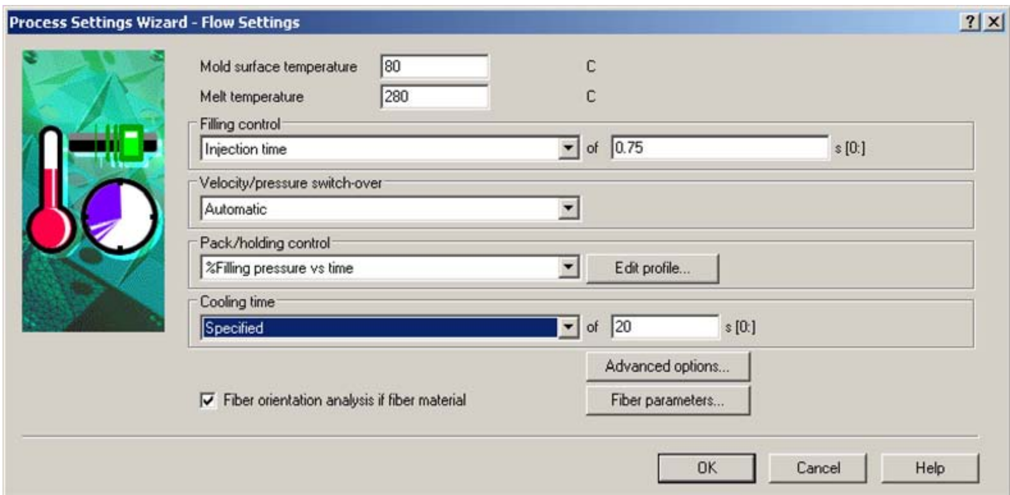


Figure 357: Process settings wizard

The fiber orientation analysis check box should be checked off if the analysis sequence is not going to go to warp or stress. The fiber analysis adds about 40% to the analysis time.

Fiber parameters

The solver parameters shown in Figure 358, are generally not adjusted unless research is being conducted. The C_i and D_z by default are automatically calculated C_i. relates to fiber to fiber interaction and D_z relates to the orientation due to the thickness of the part. These values can be automatically calculated or manually set. The fiber inlet condition at the gate can be specified. The fibers in both choices are aligned at the gate. The the center of the cross section (core), the fibers can be transverse or random. Transverse is better to use when there is radial flow in the gate area. Random is best if the flow is unidirectional and does not have a radial component to the direction. The Composite property calculation options button opens the dialog shown in Figure 359. These options are also available for research and are not generally modified.

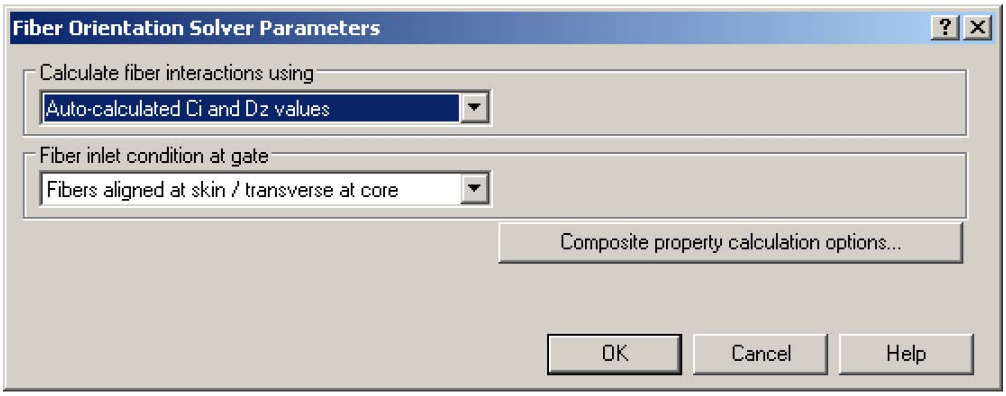


Figure 358: Fiber orientation solver parameters

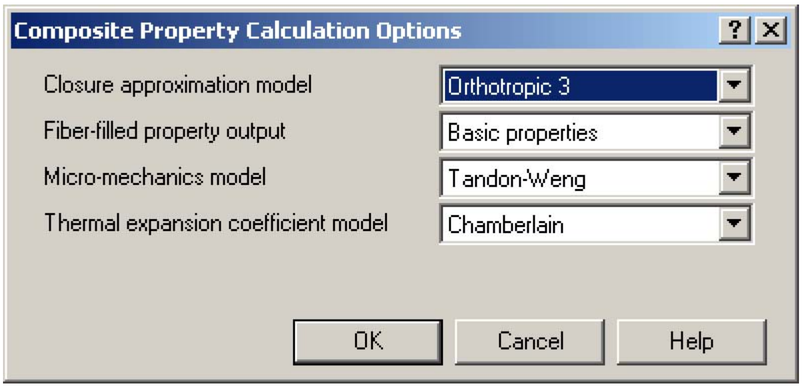


Figure 359: Composite proper calculation options

Reviewing midplane and Fusion results

There are two ways to look at the fiber orientation results, for both the Average fiber orientation and fiber orientation tensor plots. This is as a tensor component, or as a tensor principle value.

Tensor principal values

The default setting is tensor principal value, with the first principal direction. This is shown in the top plot of Figure 361. The first principal direction indicates what fraction of the fibers that are oriented in the major axis direction of the glyph shown in the element. The scale goes from approximately 0.5 to 1.0. A value of 1.0 would mean that all the fibers are aligned in the direction of the glyph. A value of 0.5 would indicate all the fibers are randomized in the plane of the element. A number below 0.5 would indicate some alignment in the thickness direction of the element.

Tensor Components

The second way is tensor components. The plot properties for tensor components are shown in Figure 360. Here there are 6 components, Txx, Tyy, Tzz, Txy, Txz, and Tyz. With tensor components, the magnitude of diagonal terms can be from about 0.0 to 1.0, while off-diagonal terms go from -0.5 to 0.5. With tensor components, the global axis direction is important. In the bottom plot in Figure 361, the results plotted are the tensor component Txx. With Txx, the value of 1.0 would indicate all the fibers are oriented in the global X direction. Comparing the plots in Figure 361, you can see that the glyphs are all oriented in the same direction for each element, but the color assigned to them is dependent on the tensor definition used. In most cases, the plot easiest to interpret is the Tensor Principal value of First.

Normally fiber orientation results are difficult to interpret. The results are normally reviewed after the warpage analysis is done and it is determined that orientation is the dominant cause of warpage. Most times with a fiber filled material, the fiber orientation dominates the cause of warpage. When this happens, the fiber orientation is reviewed. Normally you look for where the fiber orientation is NOT consistent.

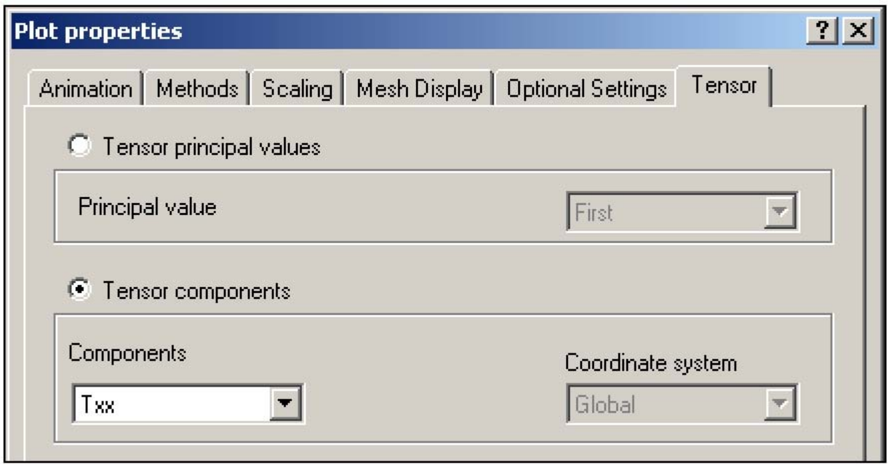


Figure 360: Fiber tensor options

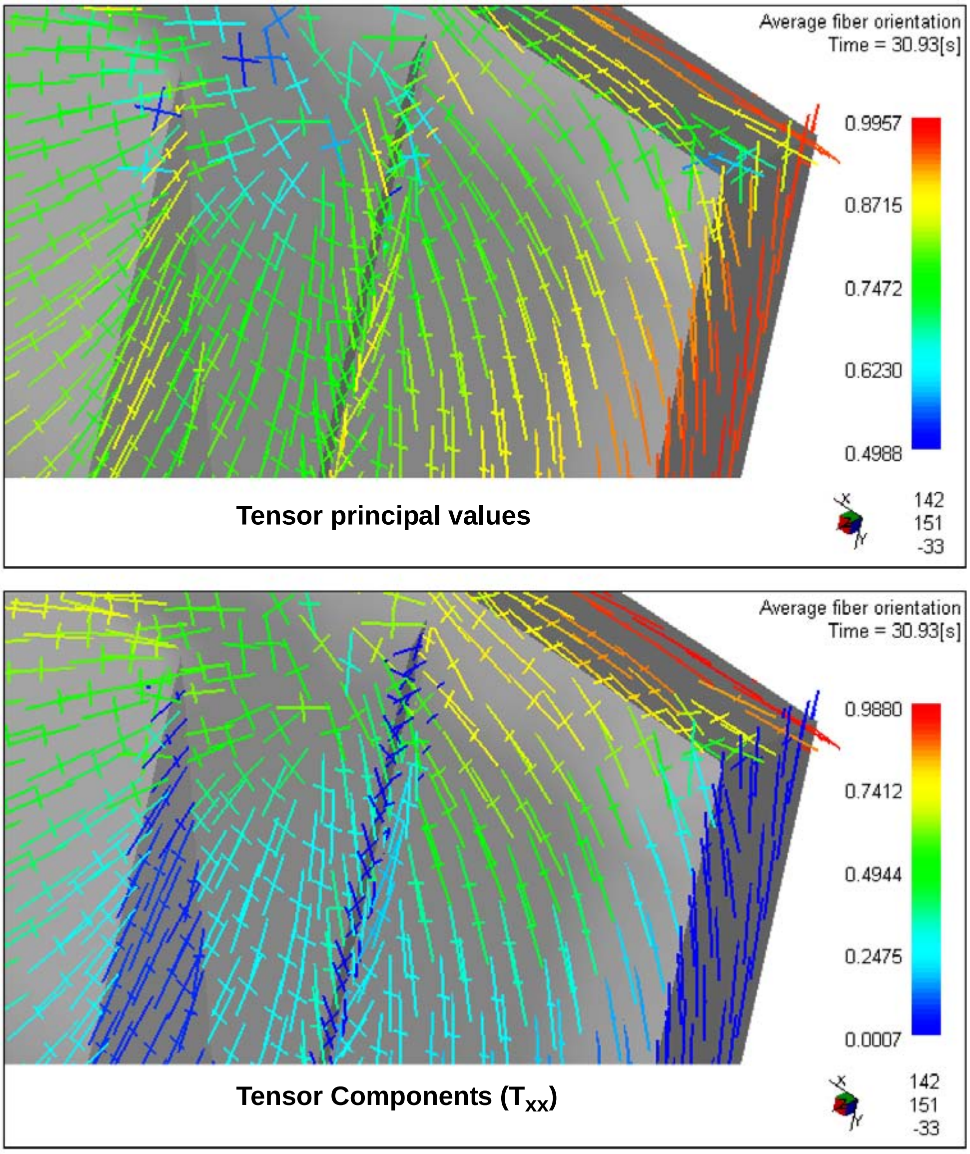


Figure 361: Tensor output variations

Interpreting 3D fiber orientation results

- 3D Fiber orientation tensors show how the fibers are oriented in an element at the end of the cycle.
- The tensor by default is plotted by **First Principal value**. This means that the highest probability percentage of the fibers align in this direction.
- The scale for principal values goes from 0.0 (blue) to 1.0 (red).
- First principal values typically range from 0.33 to 1.0
- Planar random orientation means a11 and a22 are equal to 0.5.
- Three dimensional random orientation means a11 = a22 = a33 = 0.33.

- Zero in any component means there is no percentage of fibers distributed in this direction.
- The tensor symbol, (three lines crossing) will have the legs at the same length with a random distribution, and very short secondary legs when the tensor value is high.

Figure 362 shows the fiber orientation of a part near a rib. In general, you can see how the outer laminates of the cross section have higher fiber orientation, and the center has more randomization.

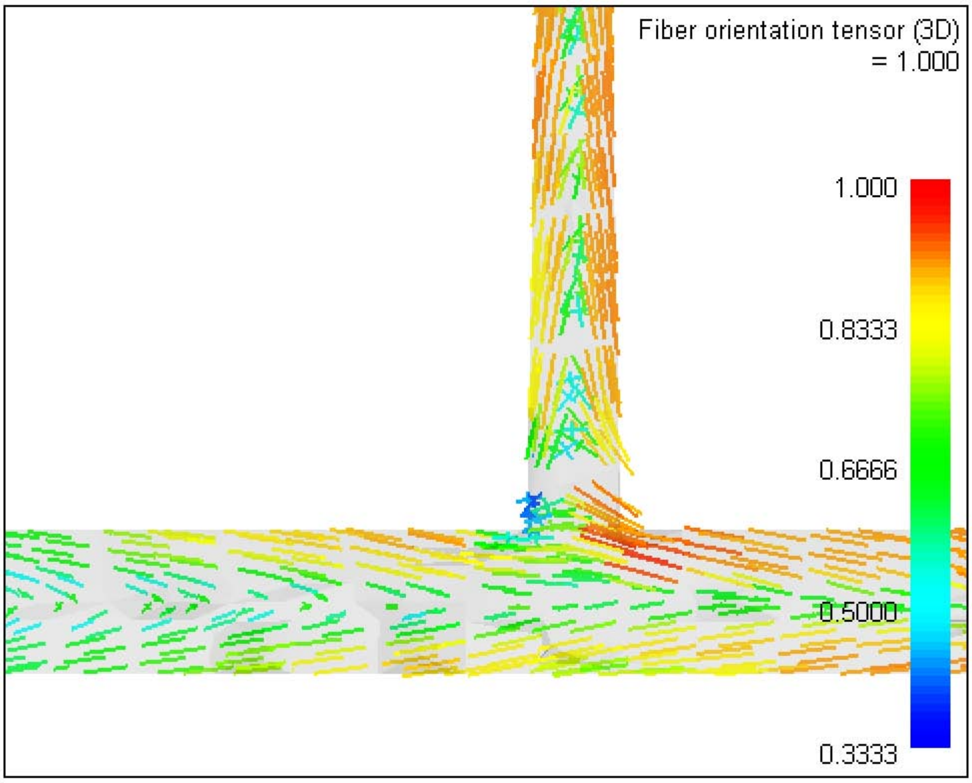


Figure 362: Fiber orientation near a rib

Useful display options

Looking at fiber orientation results can be complicated. The tensor symbols overlap by default and can get very difficult to see. Adjusting some plot properties will help with the interpretation of the results including:

- Animation tab.
  - Animate over Single Data Set (if necessary).
  - Set value range to 0.1.
  - Click Current frame only.
- Scaling tab.
  - Specified, Range to cover full range of plot i.e. 0.3 to 1 (Multiples of value range).
- Tensor tab.

- Glyph size scale factor, a low number, 0.15 to.30. Try several values to see which ones work best.

An example of using the setting is shown in Figure 363. The settings on the animation tab are done so the orientation glyph for only a limited number of elements will be displayed at any one time. With the value of 0.1 the range is typically broken down into 5 or 6 animation frames. The scaling is done so the range set up by the animation tab is a very round number. The Glyph size is changed is the glyph from one element to the next don't overlap to much.

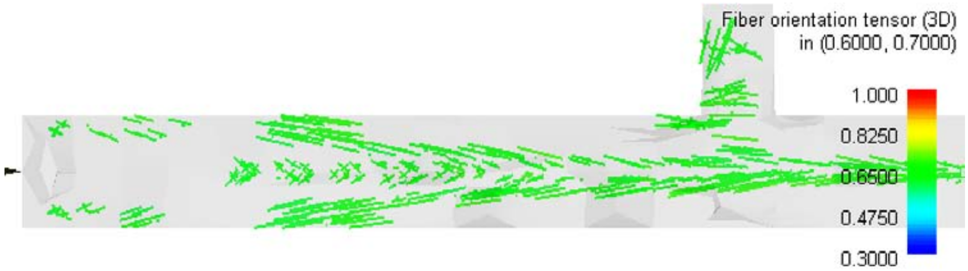


Figure 363: Fiber orientation on a 3D part with only elements with an orientation between 0.6 and 0.7 shown

References

[1]Advani, S.G., "Prediction of Fiber Orientation During Processing of Short Fiber Composites", Ph.D. thesis, University of Illinois, Urbarna, IL (1987)

[2]Tucker III, C.L., "Predicting Fiber Orientation in Short Fiber Composites", Department of Mechanical and Industrial engineering, University of Illinois, Urbarna, IL, pre-print received in August, 1989

What you've learned

MPI/Fiber is a flow analysis that adds the ability to predict the fiber orientation distribution within a polymer matrix. Fiber is run primarily as input to a warpage or stress analysis. Fibers are a classification of polymer fillers that have an aspect ratio of greater than 1. MPI/Fiber is designed to calculate the distribution of short fibers. Most materials on the database have an aspect ratio of 25.

The results from MPI/Fiber include:

- Average fiber orientation.
- Fiber orientation tensor.
- Poisson's ratio.
- Shear Modulus.
- Tensile Modulus, 1st and 2nd principle directions.
- Linear Thermal Expansion Coefficient, 1st and 2nd principle directions.

Average fiber orientation and Fiber orientation tensor are results that can be viewed to look at how fibers are aligned in the polymer. The others are primary created for import into stress and warp solvers.

The orientation of fibers are defined by an orientation ellipsoid. With this ellipsoid, the orientation is defined either as principle values, or by coordinate values.

When running a fiber orientation analysis, all the mechanical properties for both the polymer and fiber are defined in the material database. One major assumption of the solver is the fiber is short and fibers are evenly distributed in the polymer cross section.

CHAPTER 16

Results Interpretation

Aim

In this chapter you will review the types of results available from a flow analysis, how they can be manipulated and a basic interpretation of the result.

Why do it

Understanding the results from a flow analysis is one of the most important skills an analyst must possess to determine and solve problems using simulation.

Overview

We will first look at the types of results that are available. Each type of result has a default way the result can be plotted and will have different options for manipulating the results. The properties for the result allow us to change how the result can be manipulated. This will be reviewed. Finally, common results from a flow analysis will be discussed to understand its interpretation.

Review of Fusion and midplane models is very similar. 3D results provide considerable more flexibility in how the results are displayed. Examples of viewing Fusion midplane and 3D results will be discussed.

Theory and Concepts - Results Interpretation

Types of results

Results from a flow or fill analysis are available in several different forms. Depending on the type of result, there may be many ways to view that result or even change the form of the result. The forms available depend on the mesh type. Table 53 summarizes the different results available.

Table 53: Summary of result types

Result type	Midplane / Fusion	3D
Single dataset	✓	✓
Intermediate	✓	✓
Intermediate profiled	✓	
XY plot	✓	✓
Path Plot	✓	✓
Highlight	✓	
Summary	✓	✓
Probe Plot		✓

The type of results and examples of these results are listed below based upon mesh type. The lists are not comprehensive, but do show the most commonly used results.

Single dataset

Single dataset results are results that only store a single value during the filling or packing phase. The result may be stored as a nodal or elemental value. The animation of the result is from the minimum to maximum of whatever unit the result is in. Table 54 lists the single dataset results below:

Table 54: Single dataset results

Result	Midplane / Fusion	3D
Fill time	✓	✓
Pressure at V/P switchover	✓	
Temperature at flow front	✓	✓
Volumetric shrinkage at ejection	✓	
Time to freeze	✓	
Bulk temperature at end of fill	✓	
Frozen layer fraction at end of fill	✓	
Grow from	✓	
Pressure at end of fill	✓	
Sink Index	✓	

Table 54: Single dataset results

Result	Midplane / Fusion	3D
Throughput	✓	

Intermediate

Intermediate results are results that record values several times during the filling and packing phases. The default and maximum settings are listed below by mesh type. The numbers for 3D are low because the amount of disk space required. For Midplane and Fusion, The selection is done with a pull down list so the choices are limited. With 3D, any number up to the maximum can be entered. Table 55 summarizes the settings for intermediate results. These settings can be changed by using **Process Settings Wizard ➡ Advanced options ➡ Solver parameters ➡ Intermediate Output** (MP or Fusion) or **Intermediate results** (3D).

Intermediate results are animated by time. At each time step, the result is shown for the portion of the model that is full. The default scale is from the minimum to maximum seen over the entire time range. Intermediate results can also be set to display as a single data set result. This is done by specifying the time to be plotted. The result is then animated through the minimum to maximum of the unit of the result. Intermediate results are shown in Table 56.

Table 55: Settings for intermediate results

Setting	Midplane / Fusion	3D
Default Filling	20	5
Default Packing	20	5
Maximum Filling	100	50
Maximum Packing	100	50

Table 56: Intermediate results

Result	Midplane / Fusion	3D
Bulk temperature	✓	
Shear rate bulk	✓	
Frozen layer fraction	✓	
Average fiber orientation (Fiber analysis only)	✓	
Average velocity	✓	
Flow Rate, beams	✓	✓
Pressure	✓	✓
Shear stress at wall	✓	
Volumetric shrinkage	✓	✓
Density		✓
Extension rate (3D)		✓
Freeze time	✓	✓
Shear rate (3D)		✓

Table 56: Intermediate results

Result	Midplane / Fusion	3D
Shear rate, maximum		✓
Temperature (3D)		✓
Velocity (3D)		✓
Viscosity		✓

Intermediate profiled

Intermediate profiled results are only available in midplane and Fusion. Profiled results store values through the thickness of the cross section. Most profiled results are also intermediate results; therefore, the cross-sectional information is also stored several times through the filling and packing. Table 57 indicates the settings for intermediate profiles results. These settings are changes in the same location as intermediate results.

Table 57: Settings for intermediate profiled results

Setting	Midplane / Fusion	3D
Default Filling	0	N/A
Default Packing	0	N/A
Maximum Filling	20	N/A
Maximum Packing	20	N/A

The thickness direction is normalized for the result. It is referred to as normalized thickness and these values can go from 0 to 1 or –1 to 1, as shown in Figure 367. The range used depends on the type of flow analysis, **symmetric** or **asymmetric**. A symmetric analysis assumes uniform mold temperatures. This would be the case when the mold temperature used is the value entered in the flow process settings page. If a cooling analysis was run and used as input, for a flow analysis, the mold temperatures would NOT be symmetric so the asymmetric analysis is run. When a symmetric analysis is used, the normalized thicknesses go from 0 in the center to 1 at the plastic/metal interface. Then the analysis is asymmetric, the range is from –1 to 1.

For a midplane model, the –1 is the bottom side of the element, or the red side. If the model is a Fusion model, the –1 refers to the matched element to the element you are looking at, or have picked. The value of 1 then is the top side of the element for midplane, and the element you are looking at or have picked with a Fusion model.

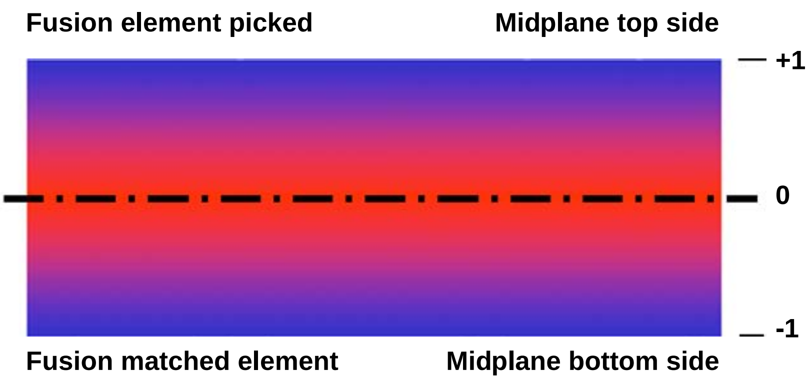


Figure 367: Normalized thickness locations

For intermediate profiled results, the initial animation is through time at a normalized thickness of zero. Most of the time, this default is modified. Animation of an intermediate profiled result can be done 3 ways.

Through time.

A value for the normalized thickness must be picked.

- Through normalized thickness.
  - A value for time must be picked.
- As a single data set.
- Both time and normalized thickness must be set.

Intermediate results that are available are shown in Table 58.

Table 58: Intermediate profiled results

Result	Midplane / Fusion	3D
Shear rate	✓	N/A
Temperature	✓	N/A
Velocity	✓	N/A
Fiber orientation Tensor (Fiber analysis and profiled only)	✓	N/A
Flow Rate, beams	✓	N/A

XY plot

An XY plot is a 2D graph. Any result with more than one value stored during the cycle and can be plotted as an XY plot. Therefore, any intermediate or intermediate profile result can be plotted as an XY plot. Some results are XY plots by default and are listed in Table 59.

Table 59: XY plots

Result	Midplane / Fusion	3D
Clamp force: XY	✓	✓
% Shot weight: XY	✓	

Table 59: XY plots

Result	Midplane / Fusion	3D
Pressure at injection location: XY	✓	✓
Recommended ram speed: XY	✓	

To plot an intermediate or profile result as an XY plot, a new plot is required. When creating the plot, the type is specified as XY Plot. The nodes or elements to be plotted can be picked by a mouse or entered manually if the numbers are known.

Path plot

The plot path or geometry dependant plot can be created for any result of the following type:

- Single data set.
- Intermediate.
- Intermediate profiled.

Path plot results create a 2D XY graph with two or more locations on the part. It shows the result change from location to location. The X-axis of the plot can be:

- Length.
- Distance from the first entity picked.
- X-coordinate.
- Y-coordinate.
- Z-coordinate.

The first entity picked is the reference for each of these values. The length is the default X-axis. In Figure 368, the plot path shows the volumetric shrinkage on the part from a gate to the last place to fill. A larger blue marker shows the location of the first entity picked. A red line shows the path and a red marker shows the location of other entities picked. When the result being displayed in the path plot format is an intermediate or profile result, the XY plot can be animated.

All XY plot results can be exported. Using the path plot is a very useful way to output results at specific locations. This data can be imported into a spreadsheet for comparison between studies.

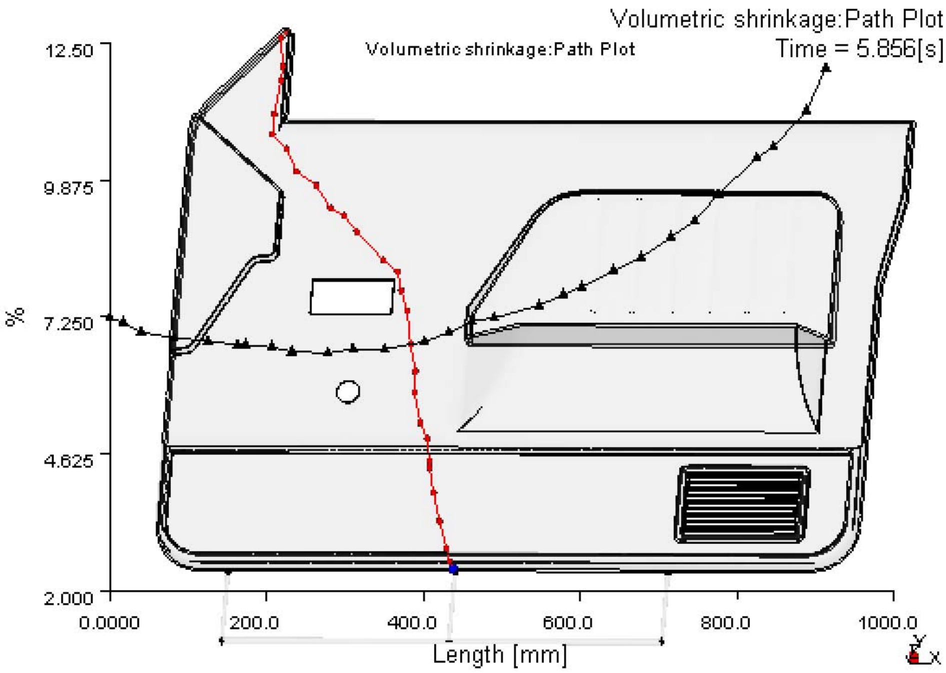


Figure 368: Path plot result

Highlight

Highlight results shown in Table 60, display a line(s) showing where the result is on the model. The weld line and air trap results are mesh density sensitive. If the quality of the mesh is not good, the result can be misleading, and therefore these results should be used in conjunction with fill time, and other results.

Table 60: Highlight results

Result	Midplane / Fusion	3D
Air traps	✓	N/A
Weld lines	✓	N/A
Clamp force centroid	✓	N/A

Summary

Summary results are text files. They are opened when the Logs box is checked at the bottom of the display window or in the study tasks list, as shown in Figure 369. The raw data used to create the files is then processed to create the files you see. Highlight results are shown in Table 61.

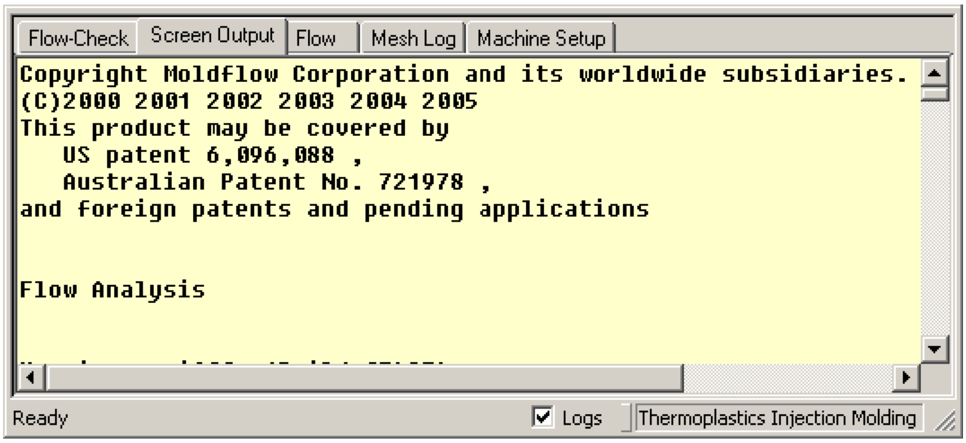


Figure 369: Summary files for a study in Synergy

Table 61: Summary results

Result	Midplane / Fusion	3D
Mesh Log	✓	✓
Analysis Check	✓	✓
Screen output	✓	✓
Results Summary	✓	✓
Machine setup	✓	✓

The screen output and results summary are very similar files. The screen output has more detail showing the actual running of the analysis. Both contain useful information about the overall analysis to get a sense for how the analysis ran and if there are any major problems.

Manipulation of results

Result creation

Most results that are viewed are predefined when the analysis is run, and are immediately available in the study task list. However, if you want an XY plot for an intermediate or profile result, a new plot must be created. The **Create New Plot** dialog can be created one of three ways including:

- Click **Results** ➡ **Create New Plot**.
- Click the New result plot icon  on the results toolbar.
- Right-click in the Study Tasks list on the Results or folder icons and select New Plot, as shown in Figure 370.

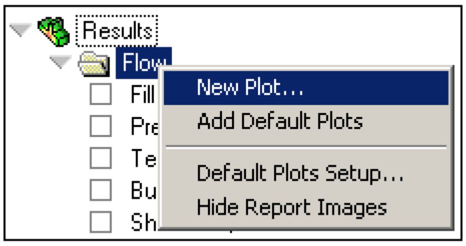


Figure 370: Create a new plot from the study tasks list

Once the Create New Plot dialog is open, shown in Figure 371, the result and plot type are chosen and the OK button is clicked to create the plot.

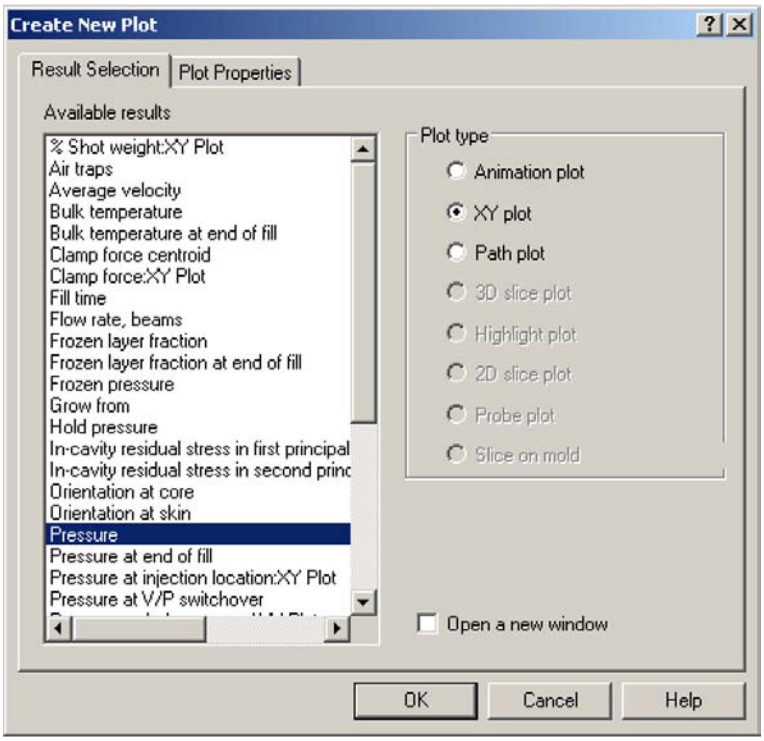
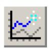


Figure 371: Create a new plot dialog

When creating an XY graph, an **Entity IDs** dialog will open on the screen and the **Add XY curve** icon  will be activated. A plus cursor  appears and this is your signal to pick locations on the model to define the entities used for the XY graph.

Plot properties

For every plot there are numerous plot properties that can be set. The options available depend on the type of result. Open plot properties by one of 3 ways:

- Right-click on the result in the study tasks list and click **Properties**.
- Click **Results** ➡ **Plot Properties**.

- Click the Plot Properties icon  on the Results toolbar.

Below is an introduction to the most commonly used plot properties.

Animation

The **Animation** tab provides access to set the type of animation if more than one type is available. In the case of Figure 372, the plot is set to Animate result over and the type is an intermediate profile result, therefore the time and normalized thickness need to be specified. If the **animate result over** field was changed to normalized thickness, a time would have to be selected to indicate at what point in time is the thickness to be animated. The opposite would be true if Time were selected for the **animate result over** field.

When the Animate result over field, shown in Figure 372, is set to **Single dataset**, the number of frames, or **Value range** can be set. By default, the number of frames is set. The value range is useful when a specific value is desired. This setting is often used with the Single dataset animation of **Current frame only**.

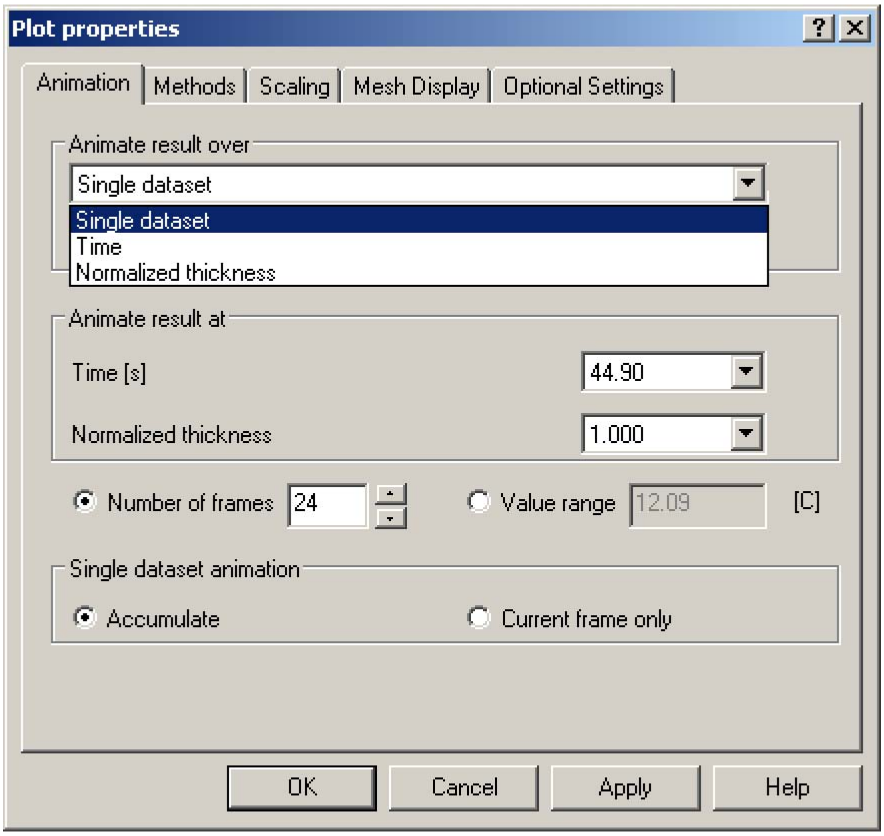


Figure 372: Plot properties, animation tab

When animating by Time or Normalized thickness, the range of the animation can be set as shown in Figure 373. This is most common for Intermediate results and limiting the plotting by time. A common range would be from the start in injection to the end of fill. Normally the scale of the results would also need to be adjusted to make best use of the change in the animation range. For example, if you want to plot bulk temperature only during the filling, the scale for bulk temperature should be changed to cover the range that occurs during filling and not the whole cycle.

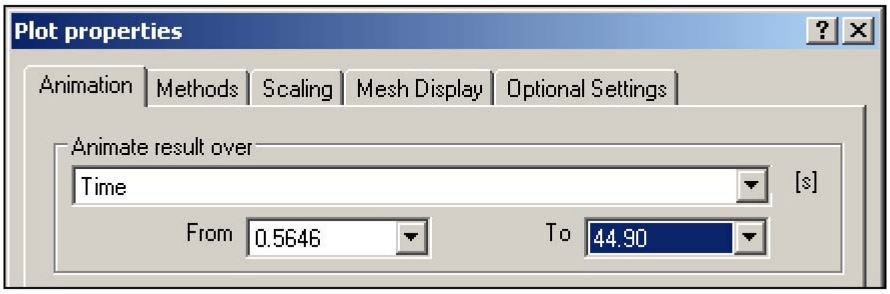


Figure 373: Animate results over

Methods

The **Methods** tab has the selection for how the result will be displayed. For the majority of results viewed, the most common options are shaded or contour. For most plots the default is shaded. The shaded option is the most "visually attractive" method, however, it is difficult to see gradients as the color shows a gradual change. Contours are individual lines that connect points of equal value, and are often used to display the fill time plot. Some results are shown as vectors or tensors rather than a shaded or contour plot. For these plots, the shaded and contour are gray and the available options are selectable.

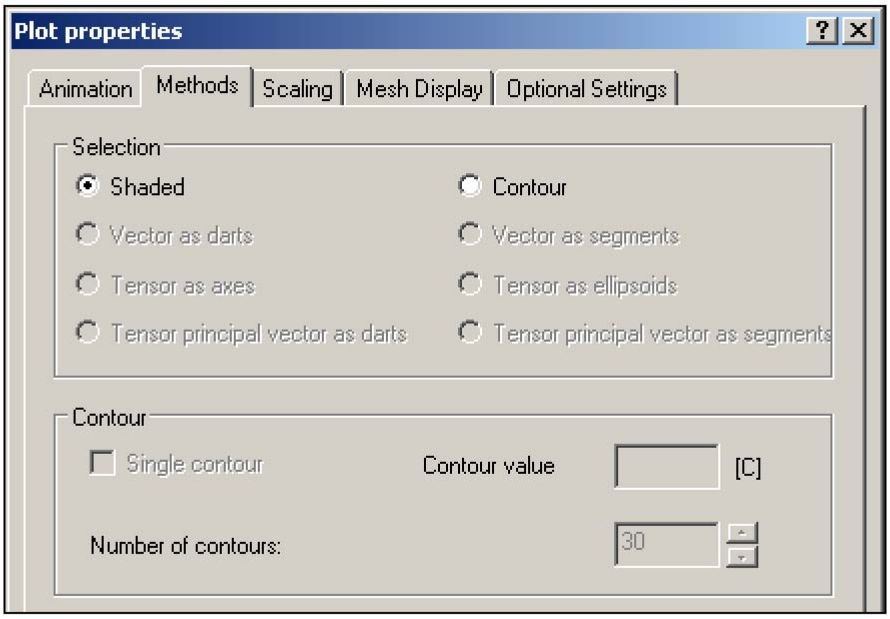


Figure 374: Plot properties, methods tab

On the Optional Settings tab, there is a banded option in the color frame that can be used with the shaded plots. This will create bands of color rather than a gradient of color. In many cases the banded color is a convenient way to show result gradients such as time or pressure. An example of the display methods is shown in Figure 374.

Figure 375: Display methods

Scaling

The **Scaling** tab shown in Figure 376, adjusts the range of the color scale on the right side of the display window. The scaling options include;

- All frames (Default).
- Per frame.
- Specified.

All frames refers to an intermediate or profile result but is used with single dataset results too. For example, with an intermediate result, it will scale each time step frame to the minimum to maximum found in all the time frames.

Per frame sets the scale differently for every time step or frame, to the maximum and minimum for that time step.

Specified allows you to enter the maximum and minimum specifically. There is another option with specified; this is **Extended color**. This will plot any area that is out of the set scale to the appropriate color, either red or blue, depending if the values are above or below the specified range.

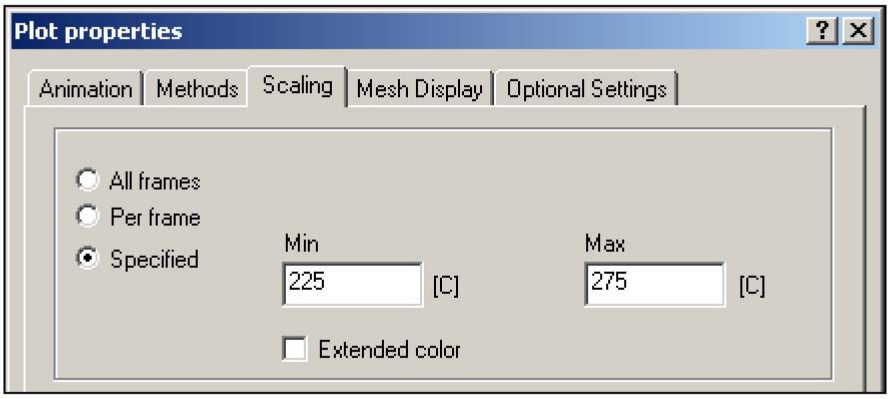


Figure 376: Plot properties, scaling tab

Mesh display

On the **Mesh Display** tab, shown in Figure 377, the appearance of the background part can be changed. The defaults change depending on the type of plot. The first section is the edge display. For most plots, the default is off. For some plots, the feature lines can be turned on to see the part geometry better. In some cases, the mesh or element lines are turned on. Normally this is done, when you want to investigate a mesh related issue found on the part. For example, the shape of the weld line is not what is expected, the mesh can be turned on to help understand why the weld line display is the way it is. Sometimes the filling may need to be adjusted. If **Transparent** is selected, the default is 0.1, which can possibly be too low. Increase it to 0.2 or higher value.

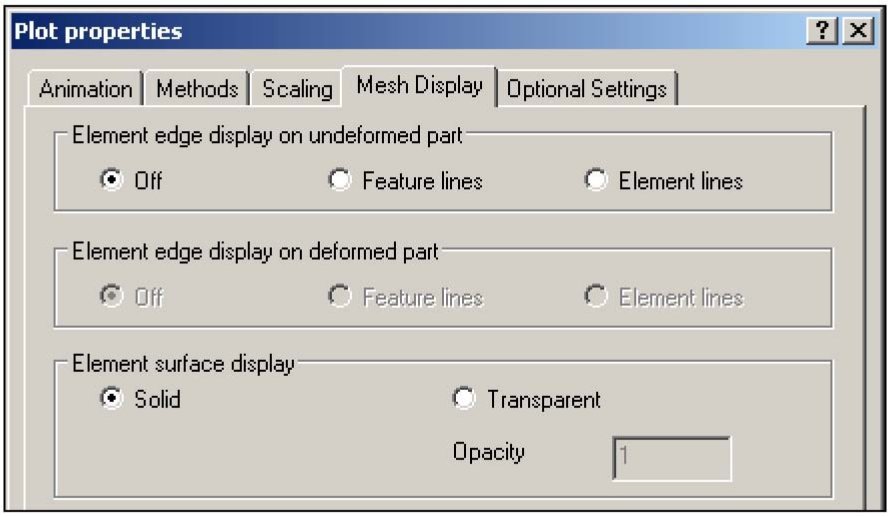


Figure 377: Plot properties, mesh display tab

Optional settings

The **Optional Settings** tab, shown in Figure 378, has several properties that can be adjusted. When discussing the methods tab the **banded** switch was mentioned. Another useful switch is the **Nodal averaged** option, and by default it is checked. For results that are stored as elemental values, such as bulk shear rate, the default display averages the elemental value to nodes. This potentially makes it difficult to see locally high values in areas such as a tapered gate. Turning off the nodal averaging allows each element to be plotted in the color that represents its value rather than being averaged out. Figure 379 shows an example of nodal averaging on (checked) and off with the Frozen layer fraction plot. With it on, the gate does not appear to be frozen. However, and the same time, the gate is plotted with the averaging off. Now the middle element is shown as frozen and the outer two do not. Turning off nodal averaging is very important when looking at results that can have significant gradients such as temperature, frozen layer fraction, shear stress and shear rate.

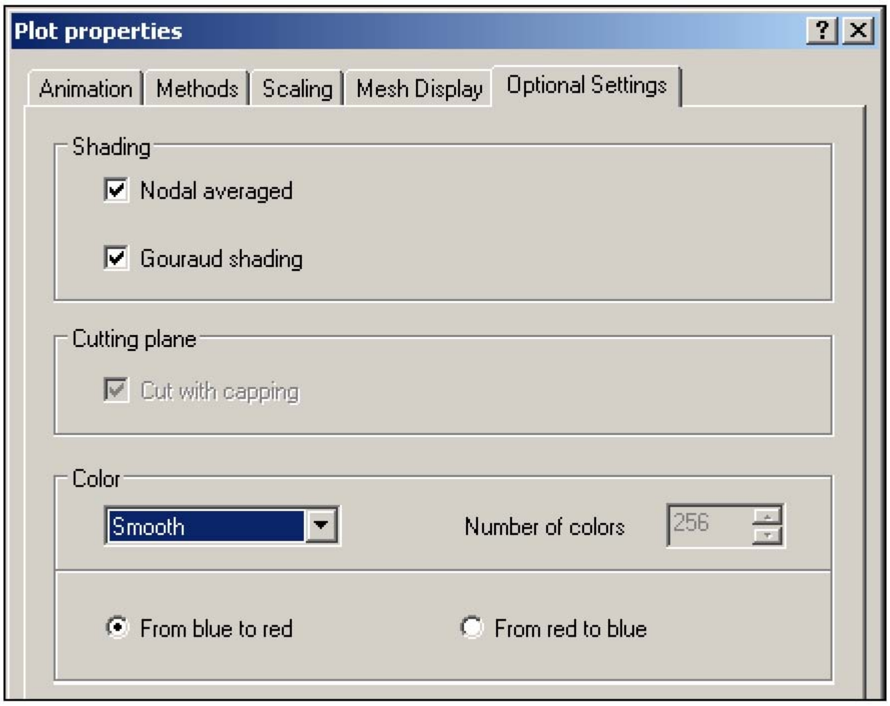
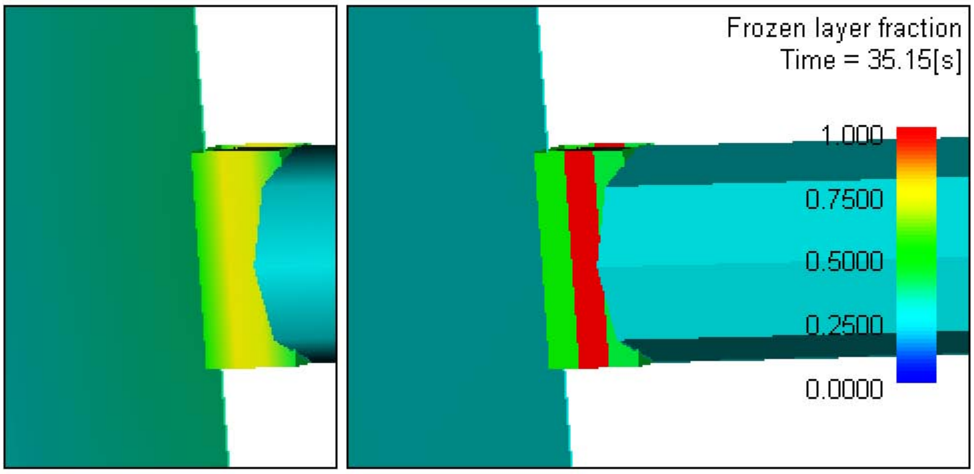


Figure 378: Plot properties, optional settings tab



Nodal averaged checked **Nodal averaged unchecked**

Figure 379: Nodal averaging

Highlight

The highlight tab will be different depending on the highlight result that is being used. For air traps and clamp force centroid, the highlight tab has a field so the color of the highlight can be changed, as shown in Figure 380. For weld lines, the highlight tab is shown in Figure 381. It has a field indicating the dataset used to display the weld line. The available datasets can be chosen by clicking the  icon. Weld lines can for example be plotted with Flow front temperature, so the temperature when the weld line is formed is shown along with the weld line itself. An example of this display is shown in Figure 382.

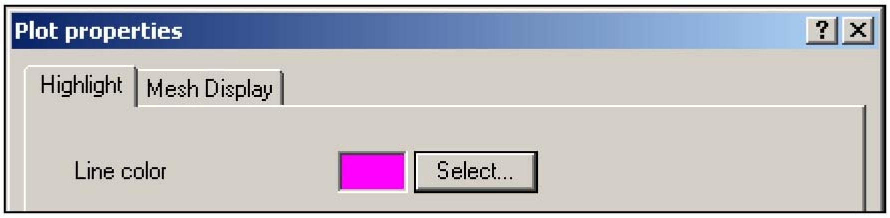


Figure 380: Plot properties, highlight tab for weld lines

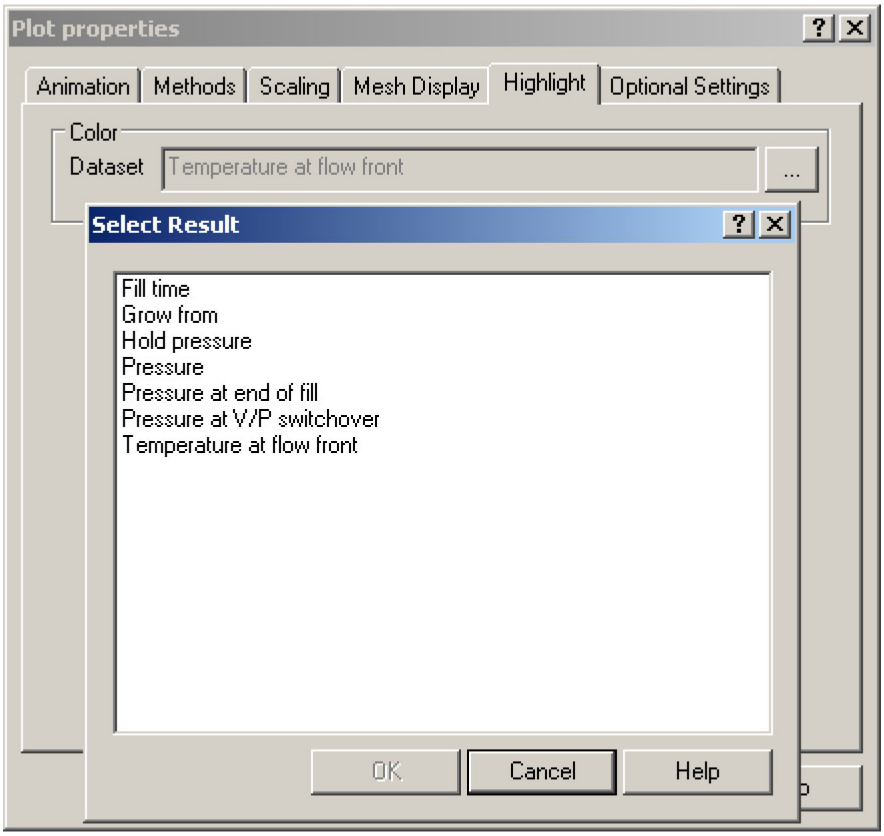


Figure 381: Plot properties, highlight tab for weld lines

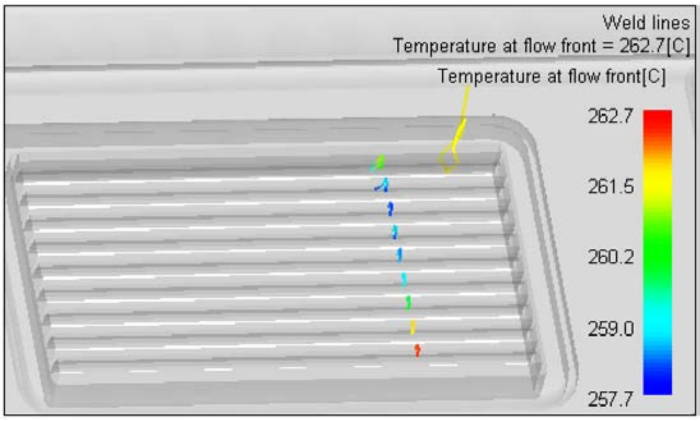


Figure 382: Weld lines plotted with flow front temperature

XY plot properties (1)

When plotting results in 2D XY plots, there are some special properties. The first tab sets what is being plotted. In the case of Figure 383 below, the independent variable is time. Normally independent variables are plotted in the X-axis but this can be changed. The number of curves and the element or node locations can also be defined. The legend location can be adjusted and the features of the plot to be displayed can be set. Moving the legend is useful so an overlay can be created between an XY plot and a shaded plot, and the legend can be positioned so it does not overlap the shaded plot scale.

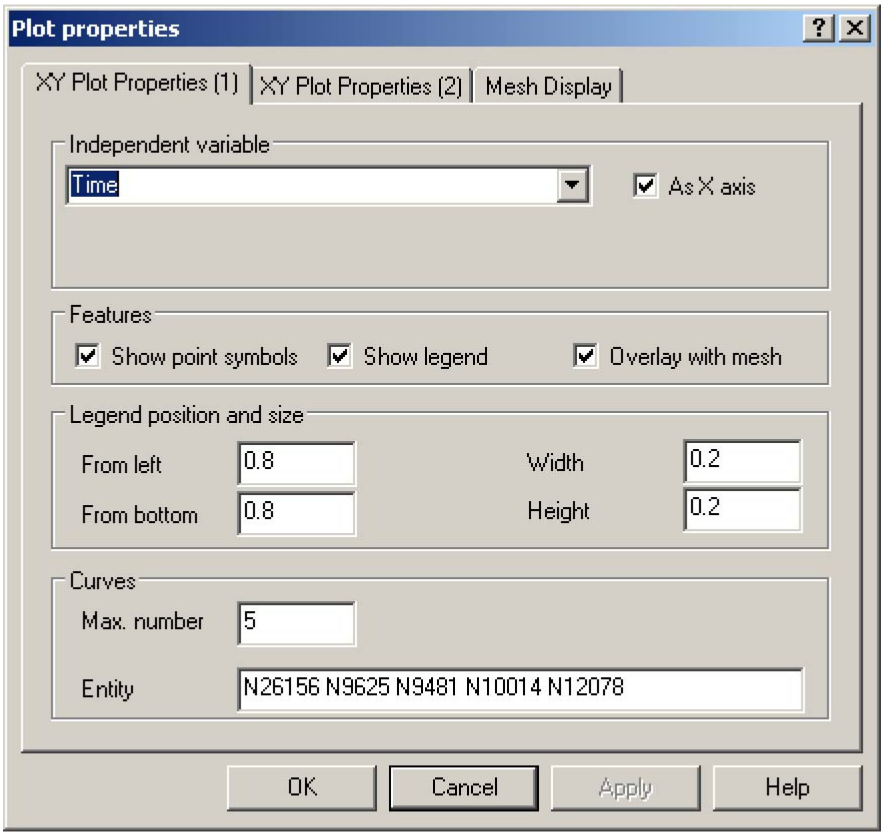


Figure 383: Plot properties, XY (1) tab

XY plot properties (2)

In Figure 384 below, XY Plot Properties (2) controls the scaling for the plot. The X and Y-axes can be scaled automatically, or can be scaled manually. In many instances, XY graphs are animated. By default, the scale changes at each time step of the animation. Manually setting the scale will ensure that each time step of the animation will be on the same page. Titles on the graph can also be changed on this page.

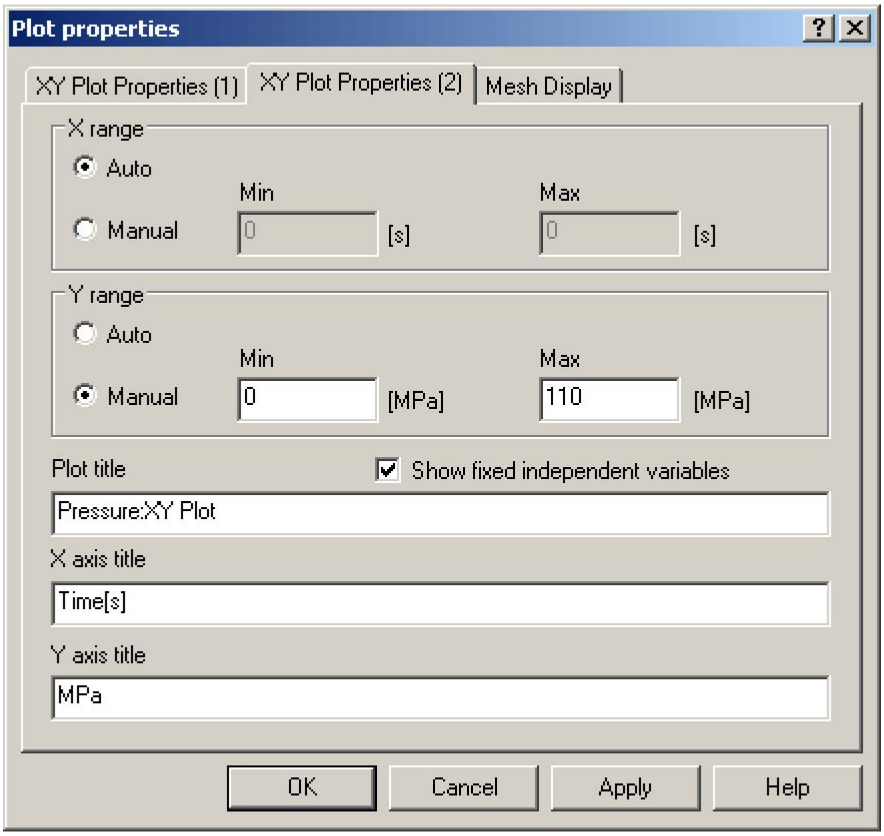


Figure 384: Plot properties, XY (2) tab

Deflection

For warpage results, there is a Deflections tab, as shown in Figure 385. This tab has two main areas, the Color frame and scale factor frame. On the Color frame, there are four fields described in Table 62.

Table 62: Options for the component property of a deflection plot

Field	Options	Description
Dataset	• All contour plots.	Default is the deflection plot being modified. This option allows a result such as volumetric shrinkage to be plotted over the deflected shape defined by the result being edited.
Magnitude/ Component	With Cartesian. • X-coordinate. • Y-coordinate. • Z-coordinate. With Cylindrical. • Radian. • Angular. • Z-coordinate.	Magnitude is the **deflection** plot. This is the total displacement of a node from it's original location to it's new location. The components available will depend on the coordinate system type.

Table 62: Options for the component property of a deflection plot

Field	Options	Description
Reference coordinate system	• Global (default). • Active anchor. • Any defined local coordinate system (LCS).	The reference coordinate systems available will depend on what has been defined. If an anchor has been defined and is active, the global is not available. If one or more LCS's have been defined, they are available and can be selected in addition to the anchor or global.
Coordinate system type	• Cartesian. • Cylindrical.	The cartesian system is used to look at linear dimensions and the cylindrical system is used to look at round features or parts to determine how the radius has changed.

With the scale factor frame, the value and direction of the warpage result exaggeration can be set. Often times it is necessary to exaggerate the deflection so it can be understood better.

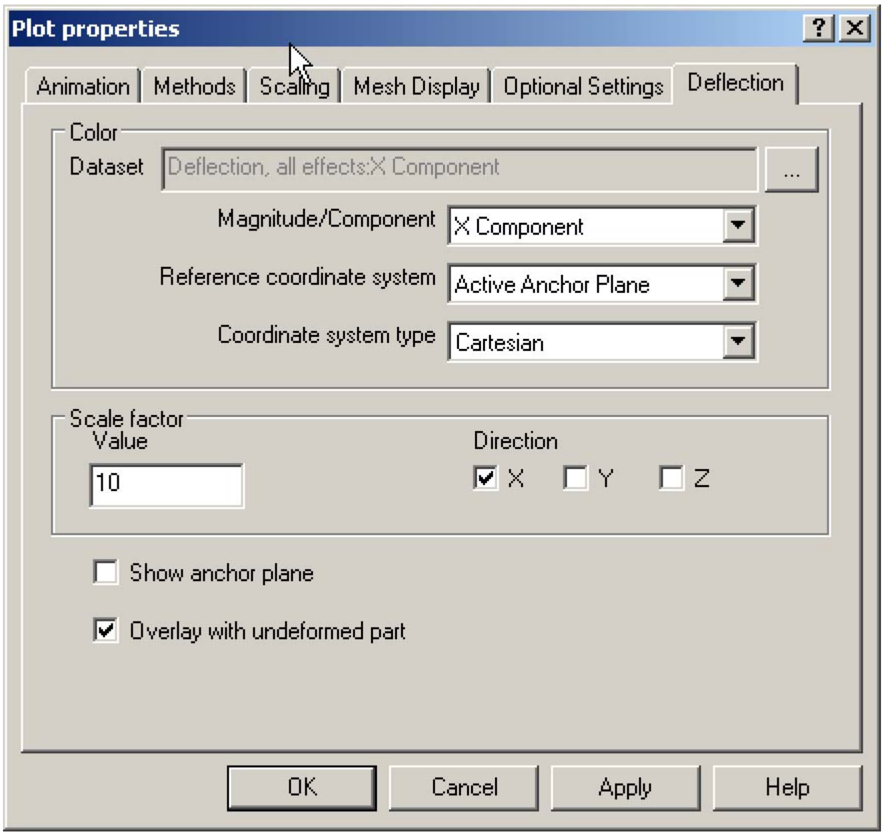


Figure 385: Plot properties, deflection tab

In Figure 386, there are 3 examples of how the deflection results can be modified. In all cases, the scale factor was set to 50. The default scale factor is 1. This magnifies the actual deflections 50 times graphically so deflections can be seen clearly. The top example is using the default warpage visualization called **best fit**, which centers the deflections on the part. The middle example scales the deflection for the X-axis only, still with the best fit. The bottom example uses a defined anchor plane and is scaled only in the X-direction. The deflection in the last example is fixed in the lower left corner of the part. As a result, all the deflections are towards that fixed corner.

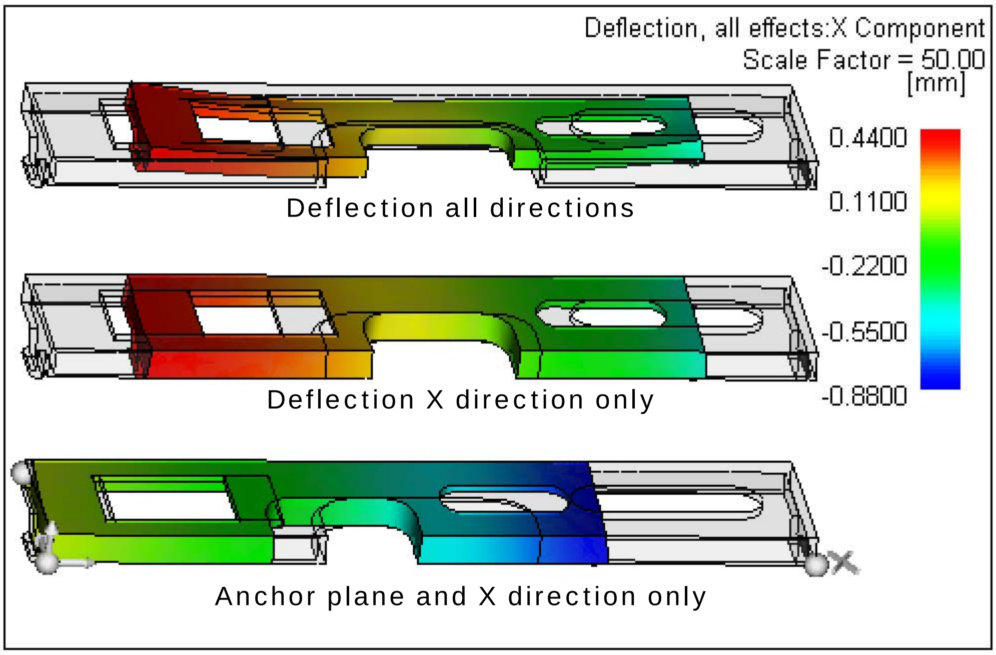


Figure 386: Deflection examples

Miscellaneous result manipulation methods

Several things can be done to aid in the display of results that does not directly involve the use of plot properties. Some of the popular manipulations are mentioned below.

Scale by layers

Visible layers automatically scale results. The Main body, and Grill layers define the scale for the plot on the left in Figure 387. The temperature range is very large from 97ºC. to 265ºC. With the plot on the right, the Grill layer is turned off. This is where all the cold areas are because of the thin walls in that area. Now the scale is much smaller from 211ºC. to 265ºC.

Using layers is a very useful tool to help scale results. If there is a particular region that is of interest, it could be handy to put it in its own layer to aid with result interpretation. Adding layers can be done after the analysis has run and saved with the model so planning ahead is not necessary.

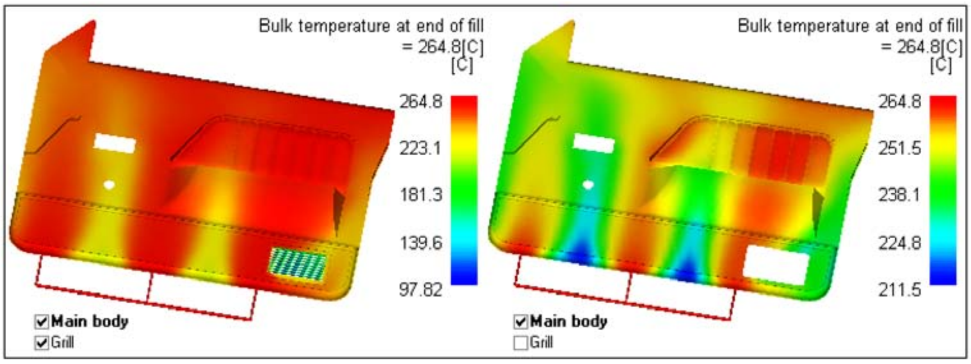


Figure 387: Scale by layers

Overlay

From the **Context menu** in Figure 388, is activated with a result name in the study task list is highlighted. One or more results can be overlaid on top of one another. It is not possible to overlay two shaded results, but a shaded and some other result can be displayed. In Figure 389, a path plot of volumetric shrinkage, and volumetric shrinkage at ejection are overlaid. The last result that is displayed has its scale displayed. Open the context menu on the result that is displayed but does not have a scale showing, and the result can be activated. If for instance, fill time and average velocity were overlaid, you may want to switch between the time and velocity scales.

The mesh display may need to be changed to a transparency of 0 or 0.1 for one of the plots to see both results.

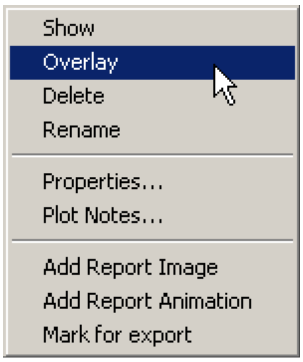


Figure 388: Context menu on a result

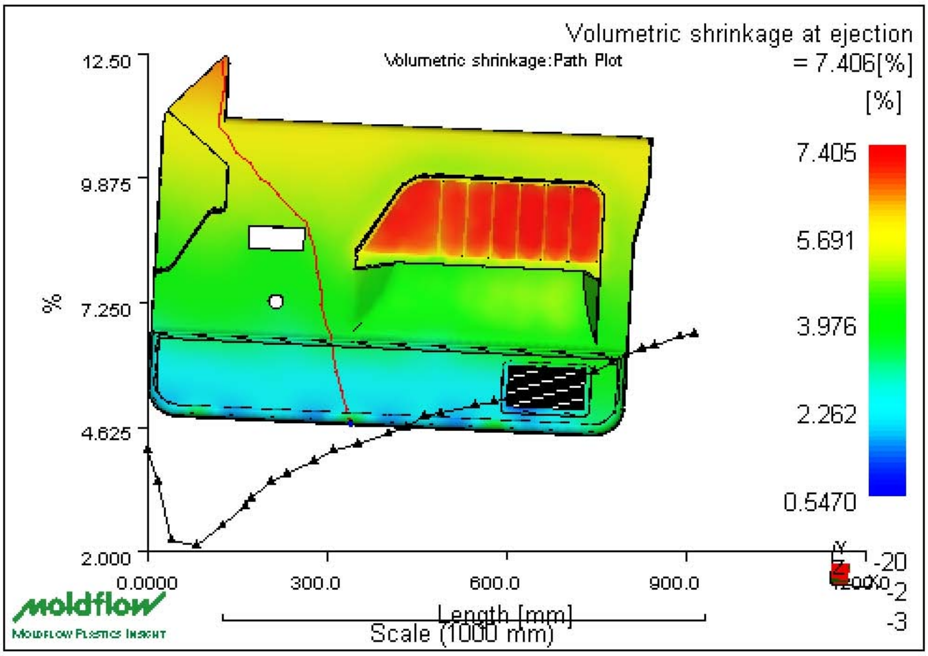


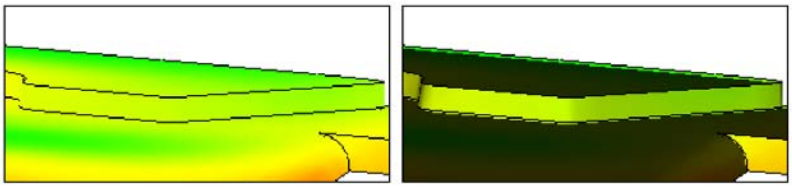
Figure 389: Overlay of volumetric shrinkage path plot and volumetric shrinkage at ejection

Lighting

In the **Preferences**, on the **Viewer** tab, there is a slider called **Lighting**, shown in Figure 390. By default, the slider is all the way to the right. This produces a highly shaded image that looks very good. However, it can make surfaces that are being viewed at low angles very dark and hard to see. Setting the slider all the way to the right does the opposite. There is no shading. The colors on the part will match the colors on the scale exactly. Some like more shading others less. For the depth perception of the results, use maximum light shading.

Figure 390: Light shading scroll bar on the viewer tab in preferences

In the example below, the same area has the maximum, and minimum shading. When looking at the part at a low angle is necessary, the maximum shading may be too high. Changing the shading can be useful to determine what level is best for easy interpretation and visually better plots. See Figure 391. For 3D results, the light shading should be set to maximum to make the results easier to interpret.



No shading Shading

Figure 391: Shading comparison on results

Cutting plane

Using cutting planes can be a useful tool to look at results in small areas of the part. By removing some of the part the small area is easier to see. In Figure 392, two predefined cutting planes were activated to look at this area. Activating and creating cutting planes is done with the Cutting planes dialog shown in Figure 393, which is opened using edit cutting planes icon . When a new cutting plane is created, the screen is the cutting plane. Once cutting planes are created, they can be moved up and down the part with the Move cutting plane command . On Fusion models, cutting planes will cut through the part so the thickness can be seen. This may help in the interpretation of the results.

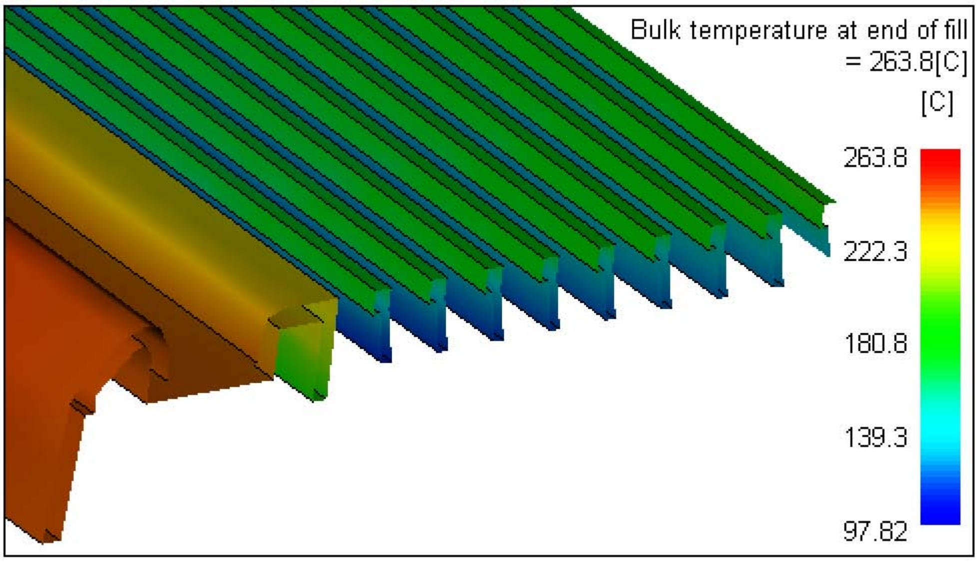


Figure 392: Cutting planes in use

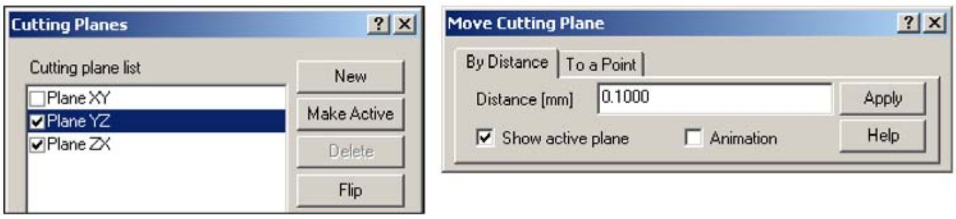


Figure 393: Dialogs for creating and moving cutting planes

Manipulation of 3D results

In the result types section above, 3D results were discussed with Midplane and Fusion. There are many results that are the same or similar between midplane/Fusion and 3D results. Also, property manipulation in many cases is similar between 3D, midplane and Fusion. This section, will discuss result manipulation that is unique to 3D.

Cutting planes

Most 3D results are best used with cutting planes. Figure 394 shows a plot of temperature. The ZX cutting plane was turned on the activated. The plane can be shown as transparent, so you know where it is. Viewing the results is best done with the cutting plane being turned off. By putting a value in the distance field then clicking the Animation button, the cutting plane will automatically scroll through the part. Uncheck the Animation box to stop the animation.

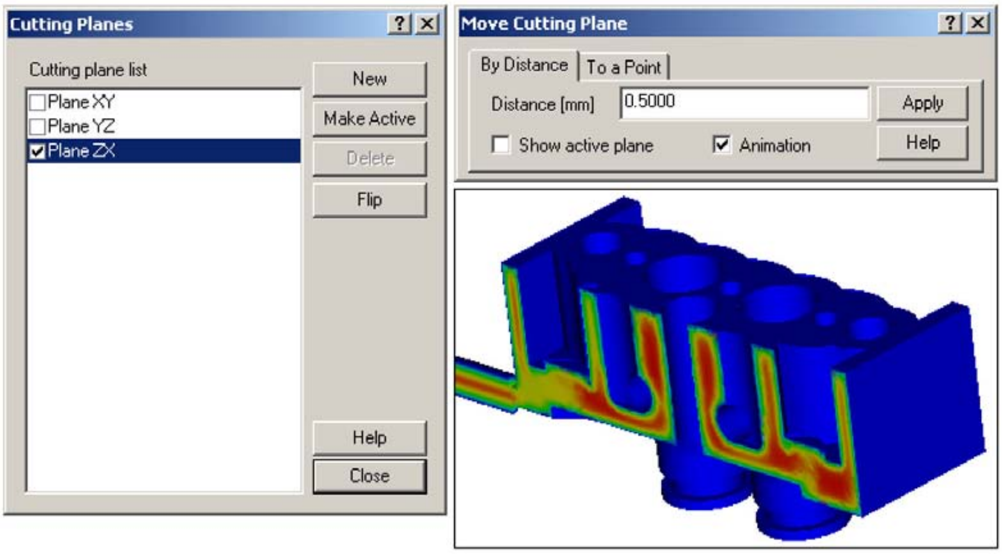


Figure 394: Manipulating results with cutting planes

Results interpretation with cutting planes

For most results, it is useful to use with a cutting plane. Results on cutting planes can be animated several ways. The cutting plane can move, or the result can be animated with the normal animation tools. For instance, Figure 395 has the Freeze time result using a cutting plane. The maximum freeze time is 66 seconds, but the plot is shown at 11.01 seconds. The areas that are not shaded (are holes) have not frozen yet. This plot is quite dramatic to watch the frozen layer grow.

Many 3D results like temperature (3D) and Volumetric shrinkage (3D), are intermediate. The resolution of the result is determined by the number of intermediate results that are written. The more results, the shorter time between results.

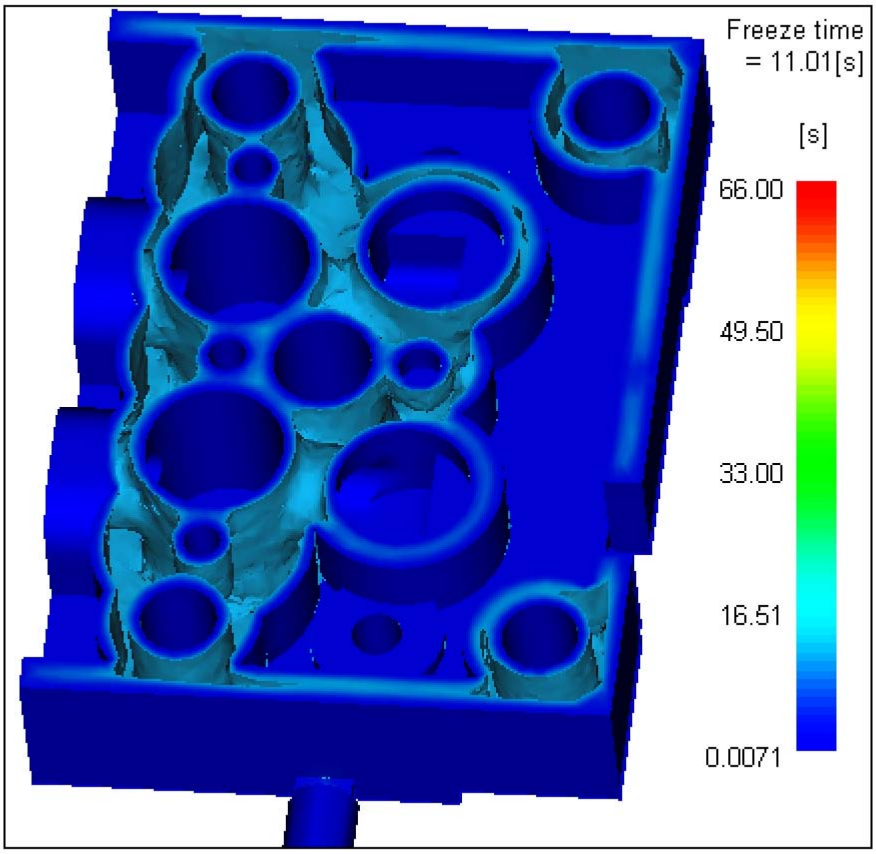


Figure 395: Freeze time results with a cutting plane

Cut with capping

On the Optional Settings tab of plot properties, there is a result called **Cut with Capping**. This is checked by default. When it is unchecked, the only result shown is on the cutting plane itself and not behind the plane, as shown in Figure 396. In some cases, this can help highlight the problem on the cutting plane and not get blended in with the part behind the plane.

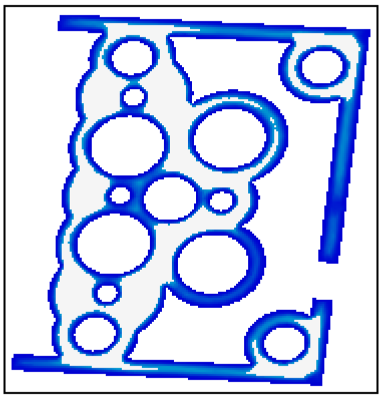


Figure 396: Same results as Figure 395, but with capping off.

XY plots

Intermediate as XY

Any intermediate result can be converted into an XY 2D plot. This is a very common tool in particular for pressure, freeze time and temperature. Many times, internal nodes are used. To pick internal nodes, the model must be separated into layers. At the layer boundary, internal nodes and elements can be seen. To assist in the node or element picking, the model is set to solid, and the element lines are added. Figure 397 shows three nodes that have been picked on internal nodes for XY plot creation.

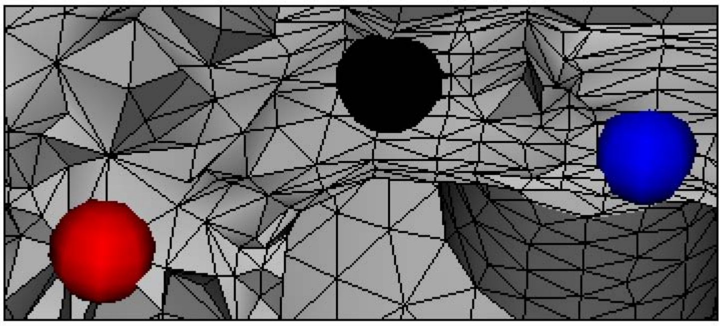


Figure 397: Internal nodes picked for an XY plot

Path plots

Any result can be created with a Path Plot type. With the path plot, the nodes or elements selected are placed into an XY graph so you can see how the variable changes at various locations in the part. If the result being used in the path plot format is an intermediate result, the result can be animated through time. If this is done, the graph looks best if the Y axis is scaled to the maximum range through all the times.

Probe plots

Probe Plot, shown in Figure 398, is only available for 3D results. Create a result by picking a location on the surface of a part. The results will show the cross sectional results for the geometry through the thickness of the part. This is an automated version of the Path plot, for results through the thickness. Probe plots can be animated, if the results are intermediate.

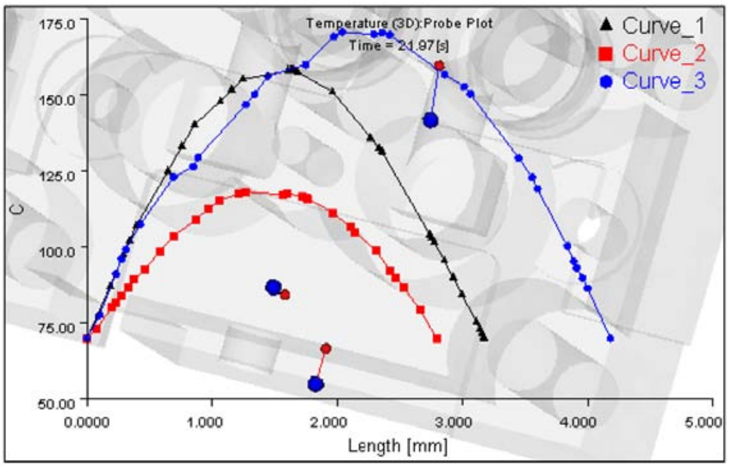


Figure 398: Probe plot of temperature

Single contours

The single contour is set on the **Methods** tab of the plot properties. Also called an iso-surface, it shows a single value for the result. Single contours are manipulated differently depending on the type of result it is.

In Figure 399 A, the result is Freeze time and is a single dataset result. Set the selection to **Contour**, and check **Single contour**. The animation is controlled by the **Number of frames** on the **Animation** tab.

The times can be controlled to round numbers. Set the scale to a round range. In Figure 399A, the Scaling tab to set the range, in this case it was set from 15 to 25 seconds, but the range could encompass the entire range for the part. Set the animation to value range and set the range to a factor of the range set on the scale tab, or set the number of frame required. In the case of Figure 399A 20 animation frames were set so when animated, the increment would be 0.5 seconds.

For intermediate results like Temperature shown in Figure 399B, a value for the iso-surface must be set. The animation is through time. In this example, the value set was 237ºC which is the transition temperature. The time the gate is frozen can be determined by using this plot. The resolution is controlled by the number of intermediate results and the length of the packing time.

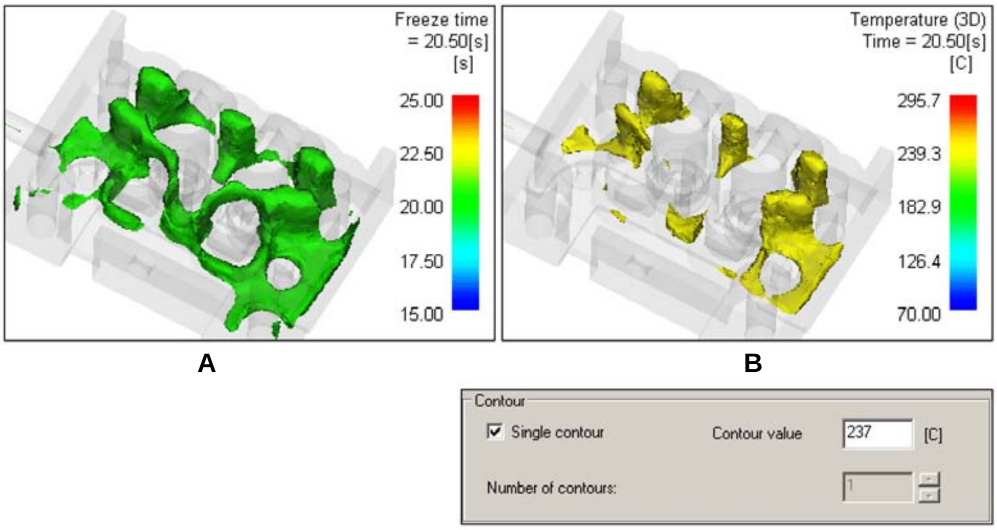


Figure 399: Single contours

General results interpretation

An important part of results interpretation is understanding the definition of the result and how to use the result. Below is a list of commonly used results with definitions and suggestions on how to use them. More commonly used results are near the top of the list.

Screen output & results summary

Midplane	Fusion	3D
✓	✓	✓

The screen output and results summary both contain information summarizing some key results of the analysis. The screen output has an additional output as shown in Figure . This can be seen as the analysis is running and provides significant information. From this you can see what the pressure and clamp tonnage are doing, what the flow rate is, and what type of control is being used.

```
  Filling phase:    Status: V  = Velocity control
                            P  = Pressure control
                            V/P= Velocity/pressure switch-over
|--------------------------------------------------------------|
|  Time   | Volume|  Pressure    | Clamp force|Flow rate|Status |
|  (s)    |  (%)  |    (MPa)     |  (tonne)   |(cm^3/s) |       |
|--------------------------------------------------------------|
|   0.23  |  2.06 |        26.79 |       6.58 |  296.12 |   V   |
|   0.45  |  5.72 |        42.93 |      18.52 |  581.55 |   V   |
|   0.68  | 10.55 |        44.51 |      29.42 |  587.23 |   V   |
|   0.90  | 15.45 |        46.07 |      47.09 |  586.74 |   V   |
|   1.13  | 20.15 |        48.55 |      82.46 |  580.91 |   V   |
|   1.35  | 24.85 |        51.35 |     127.01 |  581.82 |   V   |
|   1.58  | 29.51 |        54.46 |     182.07 |  581.39 |   V   |
|   1.81  | 34.36 |        57.85 |     249.80 |  583.03 |   V   |
|   2.03  | 38.98 |        61.01 |     321.04 |  586.03 |   V   |
|   2.26  | 43.67 |        64.07 |     401.55 |  587.49 |   V   |
|   2.48  | 48.31 |        67.42 |     502.74 |  587.12 |   V   |
|   2.70  | 52.91 |        71.27 |     627.43 |  587.51 |   V   |
|   2.93  | 57.63 |        75.16 |     757.79 |  589.25 |   V   |
|   3.16  | 62.24 |        78.98 |     892.89 |  590.75 |   V   |
|   3.38  | 66.87 |        82.15 |    1009.98 |  594.01 |   V   |
|   3.60  | 71.47 |        84.67 |    1114.70 |  595.82 |   V   |
|   3.83  | 76.20 |        87.08 |    1225.57 |  596.70 |   V   |
|   4.06  | 81.04 |        89.56 |    1348.97 |  597.30 |   V   |
|   4.28  | 85.56 |        92.34 |    1497.50 |  597.87 |   V   |
|   4.51  | 90.30 |        96.38 |    1718.66 |  598.97 |   V   |
|   4.73  | 94.85 |       100.20 |    1944.79 |  598.96 |   V   |
|   4.74  | 95.12 |       100.32 |    1960.32 |  595.54 |  V/P  |
|   4.95  | 97.84 |        92.00 |    2501.26 |  278.43 |   P   |
|   5.18  | 98.83 |        82.80 |    2737.70 |   93.42 |   P   |
|   5.24  | 99.02 |        80.26 |    2701.46 |   76.61 |   P   |
|   5.41  | 99.39 |        80.26 |    2677.81 |   73.42 |   P   |
|   5.64  | 99.76 |        80.26 |    2712.56 |   59.62 |   P   |
|   5.85  | 99.98 |        80.26 |    2767.66 |   48.62 |   P   |
|   5.86  |100.00 |        80.26 |    2776.45 |   47.78 |Filled |
|--------------------------------------------------------------|
```

Partial screen output file from a filling analysis

Both the screen output and results summary have sections similar to Figure 400. It contains information from key outputs for both during the cycle, (the first section), and at the end of fill. This can be used to quickly look at these variables to decide if any particular result should be reviewed carefully to find a problem.

```
Filling phase results summary for the part :

   Bulk temperature - maximum          (at   4.277 s) =    273.4060 C
   Bulk temperature - 95th percentile  (at   0.452 s) =    270.7860 C
   Bulk temperature - 5th percentile   (at   5.856 s) =    226.3670 C
   Bulk temperature - minimum          (at   5.846 s) =     97.8170 C

   Wall shear stress - maximum        (at   5.856 s) =      2.2689 MPa
   Wall shear stress - 95th percentile (at   4.743 s) =      0.1713 MPa

   Shear rate - maximum               (at   4.730 s) =  1.1101E+04 1/s
   Shear rate - 95th percentile       (at   0.452 s) =   5535.7500 1/s

End of filling phase results summary for the part :

   Total part weight (excluding runners)            =   2590.8999 g

   Bulk temperature - maximum                       =    263.7610 C
   Bulk temperature - 95th percentile               =    260.2170 C
   Bulk temperature - 5th percentile                =    226.3670 C
   Bulk temperature - minimum                       =     97.8220 C
   Bulk temperature - average                       =    247.3230 C
   Bulk temperature - root-mean-square deviation    =     15.7458 C

   Wall shear stress - maximum                     =      2.2689 MPa
   Wall shear stress - 95th percentile             =      0.1186 MPa
   Wall shear stress - average                     =      0.0768 MPa
   Wall shear stress - root-mean-square deviation  =      0.0861 MPa

   Frozen layer fraction - maximum                  =      0.3627
   Frozen layer fraction - 95th percentile          =      0.0415
   Frozen layer fraction - 5th percentile           =      0.0098
   Frozen layer fraction - minimum                  =      0.0000
   Frozen layer fraction - average                  =      0.0249
   Frozen layer fraction - root-mean-square deviation =      0.0168

   Shear rate - maximum                            =   1213.3199 1/s
   Shear rate - 95th percentile                    =     77.1034 1/s
   Shear rate - average                            =     25.5801 1/s
   Shear rate - root-mean-square deviation         =     35.4310 1/s
```

Figure 400: Results summary output

Fill time

Midplane	Fusion	3D
✓	✓	✓

The fill time result shows the progression of the flow front. The default plotting method is shading, but contours offer an easier interpretation of the results. The spacing of the contour should be uniform. This indicates the flow front velocity is equal. The filling of the part should be balanced. When the part is balanced, the extremities of the part fill at the same time. This is a key result that is very important in most analyses.

For 3D

The actual flow front can be seen. Fill time is used to determine the location of weld lines are interpreted on a 3D model. It's best that the light shading is set to its highest value.

Pressures

There are several different pressure plots. Each one shows the pressure of the part in a slightly different way. All the pressure plots indicate the pressure at some location at the part (a node) at some point in time.

The maximum pressure that should be used is under the limit of the molding machine. Many molding machines have a pressure limit of about 140 MPa (~20,000 psi). A good design limit for the tool is about 100 MPa, (~14,500 psi). If the molding machine to be used has a pressure limit that is higher than 140 MPa, then the design limit can go up too. The tool design limit should be about 75% of the machine limit. If the analysis does not include a feed system, the design limit should be about 50% of the machine limit.

The pressure distribution should be balanced much like the fill time. The pressure and fill time plots should look very similar. If they do, there is little or no underflow in the part.

The specific pressure result definitions are below:

Pressure

Midplane	Fusion	3D
✓	✓	✓

This is an intermediate result. Pressures at every node location through time are recorded. The default animation is time so you can see how the pressure changes through time. The distribution should be balanced, with little over packing and uniform pressure distribution during packing.

Pressure at end of fill

Midplane	Fusion	3D
✓	✓	

Pressure at the end of fill is a single dataset result. The pressure at the end of fill is normally a good plot to look at to see how balanced the part is. This plot is very sensitive to balance. If this shows a balanced distribution, the part is very well balanced.

Pressure at V/P switchover

Midplane	Fusion	3D
✓	✓	

This is a single dataset result also. This plot is also a good plot to look at the balance. Normally the highest pressure during the cycle is at switchover. The magnitude and distribution can be viewed here. Also, you can see how full the part is at switchover. The part that is not yet full is gray.

Pressure at injection location: XY Plot

Midplane	Fusion	3D
✓	✓	✓

The injection node is a common node to look at with a 2D XY plot. Looking at this plot the pressure gradient can be easily seen. There should be a steady increase in pressure once the material gets into the part. If there were spikes in pressure, generally near the end of fill, this would indicate the part is not well balanced, or the flow front velocity is speeding up due to a significant reduction in the flow front size.

Bulk temperatures

Midplane	Fusion	3D
✓	✓	

There are two bulk temperature plots and their specific definitions are listed below. Bulk temperature is a velocity weighted temperature average. The temperature of the polymer during the injection molding cycle is continually changing. It is changing not only with time, but also through the thickness. The bulk temperature captures the energy transmission within the polymer. When there is no flow, the bulk temperature is simply an average temperature of the cross section. When there is plastic flow the higher the velocity, the greater the weight that is given to that portion of the cross section.

Bulk temperature shows shear heat build up within the part. If there is significant shear the temperature will rise. The bulk temperature plots should be quite uniform in temperature during the fill. Preferably, the bulk temperature should not change more that about 5°C (~10°F) during the fill, but practically there will be larger drops. Generally up to about 20°C (~35°F) temperature drop is acceptable. If there are areas of over packing the temperature will fall significantly. This is an indication that the over packing is a problem and should be corrected if possible. Decreasing the injection time is normally the best way to narrow the bulk temperature range if it is too large.

Bulk temperature

This is an intermediate result. This is useful to see how the temperature changes over time. If a Flow analysis was run, the scale is quite large and what happens during fill is normally difficult to see. What can be done is to set the scale to per frame, watch it closely for the minimum, and maximum during filling, and then manually set the minimum, and maximum. Then animate it through the filling only.

Bulk temperature (end of filling)

This is a single data set result. The bulk temperature at the end of fill is a good snapshot of what happened during the fill. If these results look good (narrow temperature distribution) animation through the filling time is generally not necessary.

Temperature at flow front

Midplane	Fusion	3D
✓	✓	✓

For midplane and Fusion

The temperature at the flow front result is a midstream temperature when the polymer fills a node. Since the temperature is at the center of the cross section, the temperature will not change much. This plot can be used in conjunction with the weld line plot. Quality weld lines are formed at higher temperatures. The first location within a cross section where a weld line will form is at the center. If the flow front temperature is high, the weld line will generally be stronger.

For 3D

The flow front temperature is not at the midstream, but at nodes. There are many nodes through the cross section. Due to the fountain flow effect, the temperature of the polymer flow front close to the mold wall is nearly the same temperature in the center of the cross section.

Temperature

Midplane	Fusion	3D
✓	✓	✓

For midplane and Fusion

The temperature plot is an intermediate profile result. With this plot, the temperature at any location within the cross section can be seen through time or the entire cross section at a specific time. Many times an XY graph is made as specific locations to see what is happening with temperature in the cross section. XY plots can be used to see significant shear heating in the cross section. Generally, the maximum temperature within the cross section should not be above the absolute maximum temperature listed in the database.

For 3D

The temperature plot is an intermediate result. The results are most often viewed as a shaded image with a cutting plane, or as a single contour set a a critical temperature such as the transition temperature.

Shear stress at wall

Midplane	Fusion	3D
✓	✓	

The shear stress at wall is an intermediate result. At **wall** means the frozen/molten layer interface. This is the location in the cross section were the shear stress will be the highest. The shear stress within the part should be below the material limit specified in the database. Because this is an intermediate result, you can determine when the stress will be above the limit. Modifying the plot properties aids in the interpretation here. Scale the results and set the minimum value to the material limit. In this way the only elements plotted will be about the limit. Set the part to transparent. The default transparency value is 0.1; this might need to be increased, depending on the graphics card of your machine. Also it is useful to turn off nodal averaging, as this will help highlight small problem elements. Now you can manually animate through time to see when and where high shear stress occurs.

Weld lines

Midplane	Fusion	3D
✓	✓	

Weld lines occur when two flow fronts come together, or a flow front splits and comes back together, for instance around a hole. Sometimes weld lines are formed when there is significant race tracking. The material in the thick section is racing around, and the thin is lagging. At the interface between the thick and thin, a weld line may form. The weld line is sensitive to high mesh density. Due to the mesh, sometimes weld lines will be displayed that do not real exist, and other times there will be no weld line where one exists. Weld lines should be displayed with fill time to confirm their existence, and with temperature and pressure to determine their relative quality.

Reducing the number of gates can eliminate weld lines. Changing a gate location or changing wall thickness can move the location of weld lines.

Air traps

Midplane	Fusion	3D
✓	✓	✓

The air trap display will show true air traps, but also locations on the parting line where venting should occur. Like weld lines, air traps are sensitive to high mesh density. Air traps that are on parts should be eliminated. Possibly several approaches can be used. Changing the wall thickness, gate locations or injection speed normally will help eliminate air traps.

For midplane and Fusion

Air traps are lines indicating the outline and are defined at nodes when material comes to this node from all directions.

For 3D

In 3D, Air traps are defined as an iso-surface rather than a line.

Time to freeze

Midplane	Fusion	3D
✓	✓	✓

For midplane and Fusion

The time to freeze indicates the time required from the end of fill until the polymer has reached the **ejection temperature** of the polymer. This plot can be used to get an estimation of the cycle time for the part and as a starting point for how long packing needs to occur. This is also a good plot to see the effect of wide changes in wall thickness.

For 3D

The result name and definition are slightly different. It is called **Freeze time** and is measured from the start of the cycle and goes to the **ejection temperature**.

Frozen Layer fraction

Midplane	Fusion	3D
✓	✓	

There are two versions of the frozen layer fraction. The frozen layer fraction defines how thick the frozen layer is as a fraction. A value of 1.0 the cross section is entirely frozen. The reference temperature is the transition temperature.

Frozen layer fraction

This is an intermediate result. This plot is very useful to see when the part and gate freeze. If some areas of the part freeze early that are close to the gate, this could help explain why areas further from the gate have high shrinkage. Quite often, an XY plot is created at critical locations, for instance the gate.

Frozen layer fraction (end of filling)

This is a single dataset result. At the end of fill, the frozen layer fraction should not be too high. If the frozen layer fraction in some area of the part is above 0.20 to 0.25 it may suggest that packing will be difficult, and a faster injection time may be warranted. This should be used in conjunction with temperature plots to make that determination.

Volumetric shrinkage

The volumetric shrinkage indicates the volume reduction of the element expressed as a percentage due to packing of the part. The PVT characteristics play a significant roll in determining the volumetric shrinkage. The higher the packing pressure the lower the shrinkage. There are two versions of this result:

Volumetric shrinkage

Midplane	Fusion	3D
✓	✓	✓

The volumetric shrinkage result is an intermediate result. This will show how the shrinkage of the part changed through the packing and cooling stages for the part.

For 3D

For 3D, the volumetric shrinkage will show different values through the thickness of the part in addition differences across the part.

Volumetric shrinkage (at ejection)

Midplane	Fusion	3D
✓	✓	

This is a single data set result of the volumetric shrinkage at the end of fill. The shrinkage should be uniform through the cavity. It is generally not uniform. Packing profiles can be done to make the shrinkage more uniform.

Average Velocity

Midplane	Fusion	3D
✓	✓	✓

For midplane and Fusion

The average velocity shows the direction and magnitude of the average velocity in the cross section for every element through time. This plot can be very useful to see how the flow direction changes and where the higher velocities in the part are.

Most of the time, this result should be scaled. Normally the gate or some elements close to the gate will dominate the maximum value. An easy way to scale the result is to have animation running when the properties to the result are opened. Change the maximum value and hit apply. See if there is a better resolution of the velocities. Since Apply was selected, the dialog stays open. Adjust the maximum value as necessary. This is a good plot to leave the extended color on.

For 3D

The plot is a Velocity plot. This shows the velocity and direction in each element. It is an intermediate result. Since there are multiple elements through the thickness it is the velocity in each element.

Shear rate, bulk

Midplane	Fusion	3D
✓	✓	

This is the average shear rate for the entire cross-section. The bulk shear rate can be compared directly to the shear rate limit of the material as listed in the material database.

A good way to plot this result is with the nodal averaging turned off. Normally, there will be one small element that has an excessive shear rate. Turning off the nodal averaging makes it easier to see.

The shear rate is rarely too high in the part. Normally if the shear rate is too high, it will be in the feed system, usually the gate. If possible when the material has many additives, from fillers to colorants to stabilizers the shear rate should be kept well below the material limit. The additives to the polymer are generally more sensitive to shear than the polymer. If the shear rate can be kept to 20,000 1/sec. this would be great. Normally practical gate sizing considerations prevent this however.

Shear rate

Midplane	Fusion	3D
✓	✓	✓

For midplane and Fusion

Shear rate is an intermediate profile result. Most of the time, this result is viewed as an XY plot. Normally elements that have a high bulk shear rate are plotted. This will show the maximum shear rate in the cross section at a specific point in time. If the shear rate were significantly over the material limit, this would suggest there might be some problems due to the high shear. The problems could be appearance related, (gate blush for instance) or could result in lower mechanical properties of the part.

For 3D

For 3D, there are two versions.

Shear rate(3D)

This shows the shear rate in the element at the time the file was written. This result is an intermediate result.

Shear rate, maximum

This shows the maximum shear rate at the time the file was written. This result is an intermediate result.

Recommended ram speed: XY plot

Midplane	Fusion	3D
✓	✓	

A recommended ram speed profile is created that will produce a flow front velocity that is more uniform. This will help eliminate pressure spikes and may improve the surface finish of the part. The profile shown with this result can be used is a subsequent study to see the effects of the profile.

Grow from

Midplane	Fusion	3D
✓	✓	

When there are multiple gates on a part, this plot will show what triangular elements are filled from what gate. This can aid in the placement and balance of multi-gated parts.

Clamp force: XY plot

Midplane	Fusion	3D
✓	✓	✓

This XY plot shows how the clamp tonnage changes over time. The clamp tonnage is calculated based on the global XY plane as the parting plane. The clamp force is calculated in each element using the projected area on the XY plane and the pressure in that element. Fusion takes into consideration matched element sets so it does not double the clamp prediction. However, if the geometry of the part has one area that is over another relative to the XY plane, an over prediction of the clamp tonnage can occur. A property can be set to exclude elements from the clamp force calculations.

Clamp force can be very sensitive to balance of fill, pack pressure and when the V/P switchover occurs. Slight changes in these parameters can make a big difference in the clamp force.

Clamp force centroid

Midplane	Fusion	3D
✓	✓	

The clamp force centroid will mark the location of the centroid of the clamp force when the clamp force is at its maximum. This plot can be very useful if the part(s) being molded are in a press that is too small or close to the limit. If the clamp force centroid is NOT in the center of the tool, chances are the full capacity of the molding machine's clamp tonnage capacity is not being utilized. For example, if the molding machine has a 1000-ton capacity, each tie bar contributes 250 tons to the capacity. If the centroid is much closer to one or two of the tie bars than the others the effective machine capacity is lowered. This result can be used to check the overall balance of the tool. If the centroid of the tool is not in the center, corrective action can be taken.

Sink index

Midplane	Fusion	3D
✓	✓	

The sink index gives an indication for the relative likelihood of sink marks on the part. The higher the value the more likely sink marks or voids will occur. Both the volumetric shrinkage and part thickness are used in the calculation. This plot is best used as a relative tool to compare one result to another.

Velocity

Midplane	Fusion	3D
✓	✓	✓

For midplane and Fusion

This is an intermediate profiled result showing how the velocity changes through the thickness and time. Usually this is used as an XY plot to show how the velocity changes through the cross-section.

For 3D

This shows the velocity and direction in each element. It is an intermediate result.

% Shot weight: XY plot

Midplane	Fusion	3D
✓	✓	

The percent shot weight is calculated based on the part volume at room temperature density. This plot can be used to determine how the part weight will change with changes in packing time.

Summary of result types

The following table summarizes the different categories of results created by the MPI flow solvers and how to recognize them by their default animation properties.

Table 63: Results types summary

Type	Description	Default Animation Properties
Single dataset results	Distribution of values for a particular variable, for example **pressure at end of fill**, throughout the part and usually recorded at a specific time during the molding cycle.	Animation = Single dataset
Intermediate results	Distribution of values for a particular variable, for example **bulk temperature**, throughout the part at a number of different times during the molding cycle. The time value can be set in the Animation properties.	Animation = Time
Intermediate Profile results	Distribution of values for a particular variable, for example **shear rate**, across the thickness of the part at a number of times during the molding cycle. The time value or Normalized thickness value can be set in the Animation properties.	Animation = Time Independent variable = Normalized thickness
XY Plot	2D graph of any result that has multiple values i.e. intermediate and intermediate profiled results. In most cases a new plot must be created.	Animation = Usually none
Highlight	A result that shows the location of the result such as a weld line	Animation = none

What You've Learned

There are several different types of results including:

- Single dataset.
- Intermediate.
- Intermediate profiled.
- XY plot.
- Path Plot.
- Highlight.
- Summary.
- Probe Plot.

The not all types of results are available for each result and not all result types are available for all meshes. For example, "**Fill time**" is a single dataset result, while "**Temperature**" is an intermediate profiled result for midplane and Fusion. Each type of result has a different set of properties and default values for these properties. Due to the differences, the Temperature result has more flexibility in how it is manipulated.

Many results can be displayed in many different forms. A result such as shear rate, which is an intermediate profiled plot, can be displayed as like a single dataset result, an XY plot, and a path plot.

Using the plot properties to manipulate the results is often needed to efficiently interpret the results. Some of the most common properties to change include; the scale, the method: shaded or contour, and the color; smooth or banded.

Care must be used when looking at the result, as there may be subtle details about the result definition at is important, or the result definition changes slightly between 3D and the other mesh types.

CHAPTER 17

Gate & Runner Design

Aim

The aim of this chapter is to review the types of gate and runner designs, and to model multiple cavity tools using two different techniques. Finally, an automatic runner balance will be preformed to size the runners, achieving a good balance between the cavities.

Why do it

It is important to understand the different types of gates and runner systems that are available and how to model them. The two methods of creating runner systems will be used so you can see how they differ, and to learn how to do both.

Runner balancing, and sizing are important when optimizing the tool. If a runner is not sized and balanced, there are generally many problems associated with the lack of balance. Problems can include, wasting material, flash, short shots, and warpage etc.

Overview

Various gate types and how they are modeled followed by runners will discussed. You will create an eight-cavity tool using the Cavity Duplication Wizard, and manual runner creation techniques.

You will create a second runner system to represent a 2-cavity family tool using the Runner Creation Wizard. You will then size the runners using the runner balance analysis.

Theory and Concepts - Gate & Runner Design

Gate design

There are many different types of gates. For each gate type there are many variations and different names. Table 69 lists the most common types of gates. The gates listed fall into two major categories, manually trimmed gates, and automatically trimmed gates. A description and nominal dimensions of the gates is below the table. The nominal dimensions are general guidelines. These guidelines can change dramatically depending on the material and application. An excellent source of supplemental information on gate design can be obtained from the material supplier' design guide. Using simulation, gates are sized using shear rate as a guide, as discussed below.

Table 69: Gate types

Manually trimmed	Automaticlly trimmed
• Edge	• Submarine
• Tab	• Cashew
• Sprue	• Pin
• Diaphragm	• Hot drop
• Ring	• Valve
• Fan	
• Flash	

Manually trimmed gates

A manually trimmed gate is a gate that is trimmed as a secondary operation, normally by the machine operator. Some types of these gates are quite popular. However, one of their main disadvantages is that an operator needs to be dedicated to the machine to trim gates, increasing the cost of the parts.

Edge gate

Edge gates the most common type of manually trimmed gate. Figure 408 shows two variations on the edge gate, plus the mesh for an edge gate. There are many variations to this type of gate. An edge gate has a rectangular cross-section. It intersects the part at the parting or split line. The gate is normally on one side of the parting line, but can be on both sides. The cross section can be straight or it can be tapered. The dimensions of the gate includes the thickness, width, and, land length.

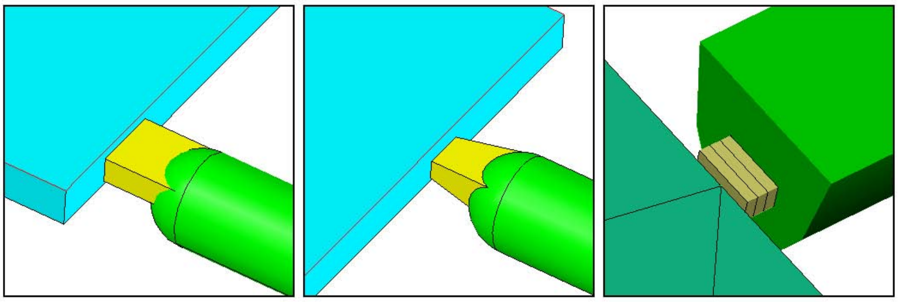
The thickness of the gate is normal to the parting line. The thickness of the gate is nominally between 25%, and 90% of the part's wall thickness. Edge gates can be as thick as the wall thickness they are entering. The larger the gate, the easier the part is to pack out and the lower the shear rate. However, ease in packing can lead to over packing of the cavity. The width of the gate is normally 2 to 4 times the gate thickness, but can be as narrow as the thickness, and as wide as 10 or more times the thickness. The land length is normally short, from under 0.25 mm to about 3.0 mm. The smaller the part is, the shorter the land length should be. The taper on the edge gate, should there be one, can be a small amount, or be tapered to be tangent with the runner on the parting line, to the runner depth, or both.

Midplane and Fusion meshes

Edge gates are normally modeled with beam elements with a rectangular cross section. The dimensions required are width and height (thickness). There should be at least 3 elements in the gate.

3D meshes

With a 3D mesh, the gate is often modeled with tetrahedral elements rather than beam elements. If this is the case, there should still be several rows of elements along the land length of the gate to properly account for shear rate, and gate freeze in particular. The runner feeding the gate can be either beams or tetrahedral elements.



Straight cross-section **Tapered cross-section** **Gate with 3 elements**

Figure 408: Edge gates

Tab gate

A tab gate is a special type of gate with a large cross-section at the tab/part interface. Figure 409 shows the tab gate and a common method of meshing. The tab is meshed with triangular elements and the gate itself is beams. Tab gates are used on parts that require very low stresses in the part such as lenses, or other optical components. There is often high stress, gate blush, and jetting associated with gates. The tab is a method to prevent these problems with the part itself. The tab is generally quite large and is normally machined off as a secondary operation. The tab prevents gate related defects and the normal edge gate freezes first defining the end of the packing time.

Midplane and Fusion meshes

The tab can be made by beam elements but it is more likely to be made with triangular elements. The edge gate should be made with at least 3 beam elements.

3D meshes

The tab should be modeled using tetrahedral elements. Like the edge gate mentioned above, the gate here can be beams or tetrahedral elements. The same is true for the runners.

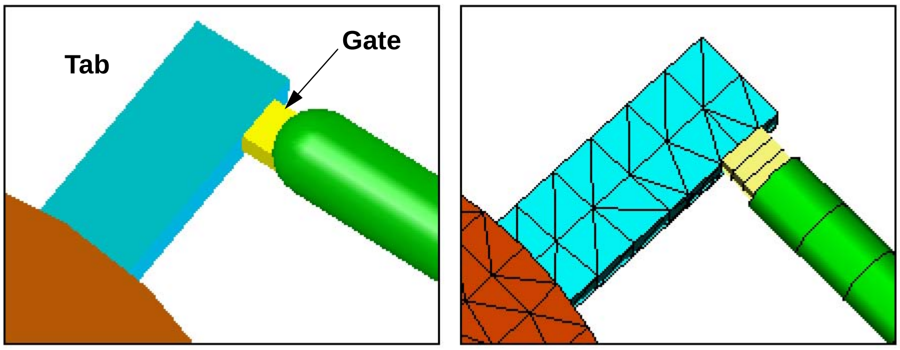


Figure 409: Tab gate

Sprue gate

A sprue gate is a normal cold sprue that is directly on the part as shown in Figure 410. This is used for larger single cavity tools. It is generally not considered a good gate design as the cross-section is much larger than the part's nominal wall thickness and it is prone to controlling the cycle time and over pack the part. The gate needs to be manually trimmed. There is also a gate mark left after trimming is very large and noticeable.

Midplane and Fusion meshes

The gate is modeled with tapered circular beam elements. Normally a curve is created with the proper orifice diameter, and the taper then meshed.

3D meshes

As with other gates for a 3D part, the sprue gate can be modeled as a beam element or with tetrahedral elements. This would be the most common way.

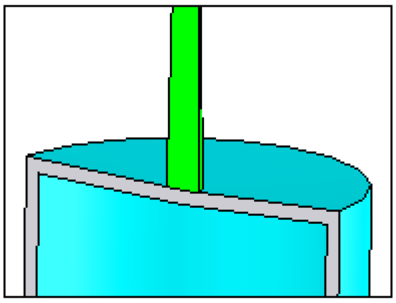


Figure 410: Sprue gate

Diaphragm gate

A diaphragm gate is a gate that goes on the inside of a cylindrical feature of a part. The purpose of this type of gate is to have a very uniform flow down the feature without any weld lines. It also produces a very straight orientation down the axis of the part. This is easily done as the gate is naturally balanced.

A diaphragm gate will have 3 main components, an entry point, body, and land. Figure 411 shows a part with the gate land and gate body. The entry point could be a sprue, a hot drop or a 3-plate mold pin gate. The body of the gate is cylindrical and connects the entry to the gate land. The body is normally relatively thick. The gate land is a narrow connection between the gate body, and the part, and is generally very thin. Because the gate land is the circumference of the cylinder, the gate can be very thin without having a high shear rate. The land is thin to aid in the trimming of the gate. This gate is expensive to use, as the de-gating often requires a secondary operation with a custom de-gating fixture.

Midplane and Fusion meshes

The entry of the gate is constructed with beam elements. The body and land are made of triangular elements. There should be at least 3 rows of elements in the thin land area to account for the hesitation and the quick freeze off that will occur in the thin land. The gate could be created as a portion of the model and imported into Synergy, or it can be constructed within Synergy. Constructing the gate within Synergy will give the best control on the mesh density but will be time consuming to model.

3D meshes

The gate land and body must be tetrahedral elements, therefore must be part of the model that is imported. The gate land should still have 3 rows of elements.

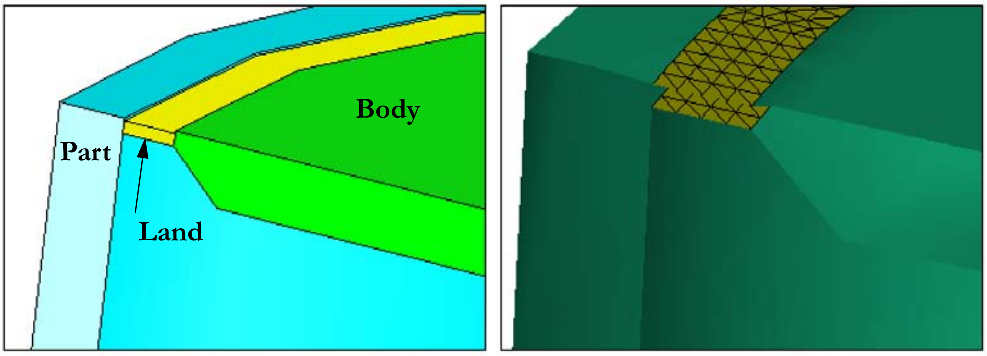


Figure 411: Diaphragm gate

Ring gate

A ring gate is a gate that goes on the outside of the cylindrical feature. Figure 412 shows an entire ring gate, a cross section and a mesh for the gate land. The objective for a ring gate is the same as a diaphragm, to produce a uniform flow front down the part. However, this gate is not naturally balanced. The gate is composed of the gate land and a runner going around the part. The material normally enters the runner at one location and has to travel 180 degrees around the part, and then flow down the part. The problem is the polymer by the entry point will flow through the land well before the material gets to the other side. If the gate land is made thinner, the polymer will hesitate and possibly freeze off at the entry point so the part will still not fill correctly. Two or three entry points can be created with even spacing, but it will still be very difficult to have a uniform flow front, and the processing window will be narrow.

Midplane and Fusion meshes

The ring runner will be constructed with beam elements and the land should be constructed with at least 3 rows of triangular elements. The easiest method of creating this type of gate is in Synergy using midplane meshes for the gate land. If the analysis is only a fill or flow analysis, a mixture Fusion for the part mesh and midplane for the gate. Beams are used to model the runner around the part.

3D meshes

A tetrahedral mesh would be the best way to model the entire gate. The gate can be modeled in the CAD system and imported with the part.

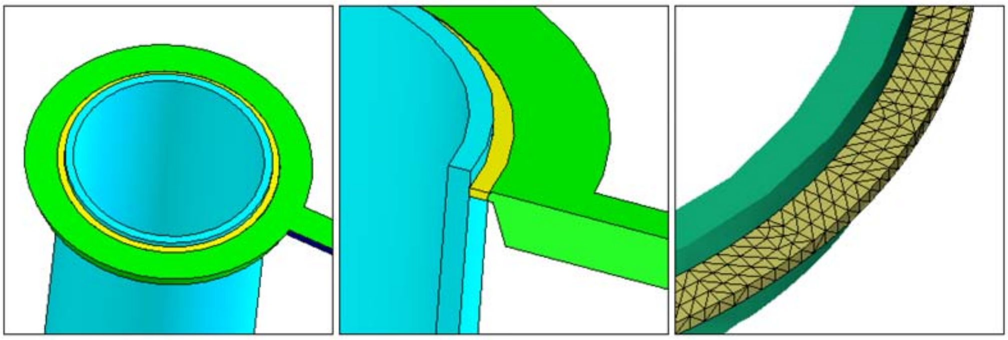


Figure 412: Ring Gate

Fan gate

A fan gate is a wide balanced edge gate. If an edge gate were made very wide with a uniform cross section the flow front entering the part would be radial. The polymer would favor the center of the gate. A fan gate is balanced so the flow front is flat. The fan gate is thinner in the middle and thicker towards the outside. A fan gate will normally have two sections, the body, and land. The land will usually be narrow and thin with a uniform cross section. The body will be triangular in shape with vertex at the end of the runner. The back edges of the fan gate, which are heavy, can be a rounded or trapezoidal in shape. In this case the back can be modeled as a beam. The thickness transition can be more gradual, and the back can be modeled with triangular elements.

Midplane and Fusion meshes

The main body of the gate should be modeled with triangular elements. The mesh density should be fairly fine to capture the thickness changes accurately. The land should be modeled with at least 3 rows of triangular elements. Since fan gates are designed to be balanced, the thickness of the fan gate may need to be altered to get the balanced flow front desired. The match ratio may be quite low on a Fusion version of the gate. The fan gate could be constructed as a midplane on a Fusion part. Remember only fill and flow analyses can then be done.

3D meshes

Fan gates such as the one shown in Figure 413 are best done in 3D to properly capture the thickness changes.

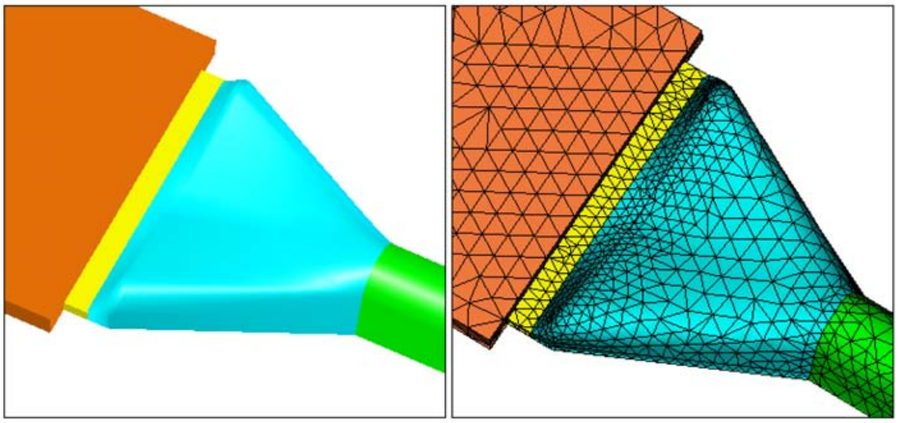


Figure 413: Fan Gate

Flash gate

A flash gate is similar to a ring gate. There is a runner parallel to the edge of a part with a thin land (flash) between the runner and the part. Flash gates are used for the same reason fan gates are; to produce a flat flow front. Flash gates suffer with the same problem the ring gate does, hesitation. The polymer starts in the center of flash gate and works its way to the end. However at the same time the material is flowing over the flash land creating a radial fill in the part. If the land is too thin and hesitates too much, the part can back fill, causing weld lines and air traps.

Midplane and Fusion meshes

The flash gate's runner will be constructed with beam elements and the land should be constructed with at least 3 rows of triangular elements. The easiest method of creating this type of gate is in Synergy using midplane meshes for the gate land. If the analysis is only a fill or flow analysis, a mixture Fusion for the part mesh and midplane for the gate. Beams are used to model the runner around the part.

3D meshes

A tetrahedral mesh shown in Figure 414, would be the best way to model the entire gate. The gate can be modeled in the CAD system and imported with the part.

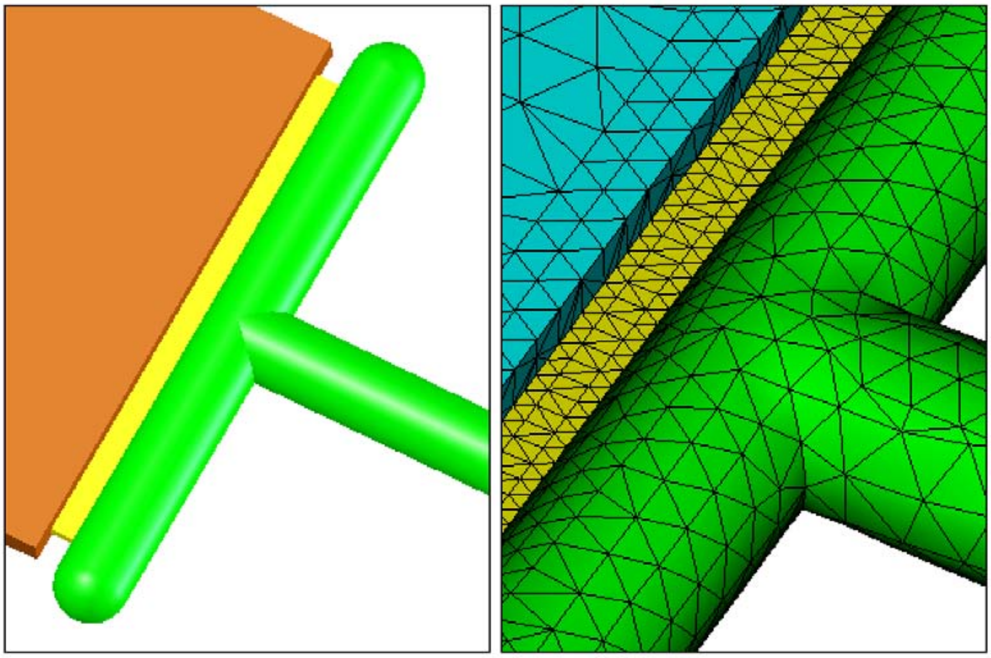


Figure 414: Flash gate

Automatically trimmed gates

Automatically trimmed gates are detached from the part as part of the ejection process. They are very widely used because there does not need to be an operator to de-gate the part, saving time and money.

Submarine gate

A submarine gate, also commonly called a tunnel gate, or sub gate, is the most common cold automatically trimmed gate. See Figure 415. The gate has a tapered cylindrical shape. The most critical dimension is the orifice diameter. This is the opening at the part. Other important dimensions include the angle to the mold face, and the included angle of the gate.

The orifice diameter is normally 50% to 75% of the wall it is going into, but it can be even larger than the part's wall thickness. The larger the orifice, the larger the witness mark that remains on the part. If the gate is too big, it can even rip a portion of the part away. The included angle is generally 10º to 20º, but can be smaller or larger depending on the situation. Many times a sub gate is sized by the orifice diameter on one side and the same diameter as the runner at the gate entrance. When this is done the included angle is normally large. The angle to mold face ranges between 30º, and 60º. A 45º angle to mold face is most common. The stiffer the material, the higher the angle should be, as the polymer has to be deformed upon ejection.

Midplane and Fusion meshes

There should be at minimum 3 elements defining the sub gate, preferably more. Since the gate is tapered, a better freeze time prediction will be achieved with more elements.

3D meshes

Sub gates can easily be modeled with beam elements for a part that has a 3D mesh. Tetrahedral elements can be used and with enough mesh density the shear rate and pressure drop will be more accurate. Meshing round features tends to take many elements. You must make sure there are still several rows of elements down the length of the gate.

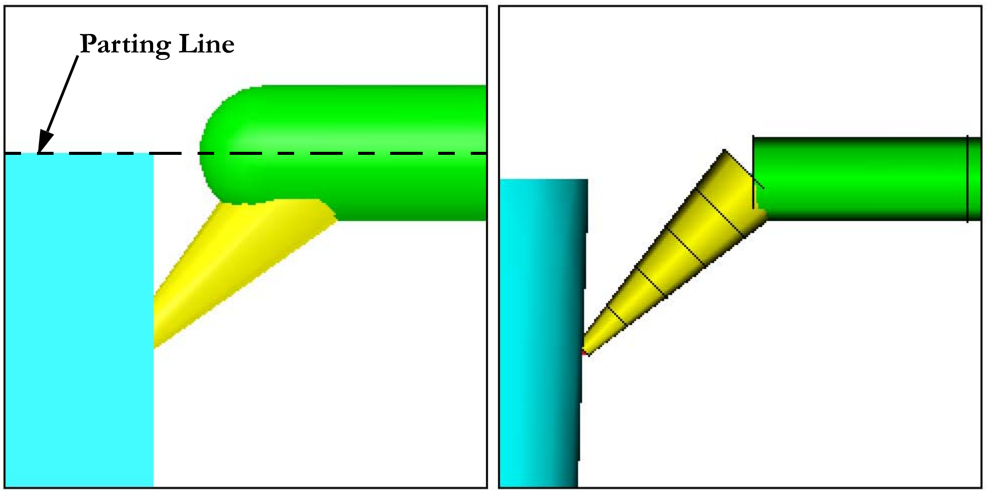


Figure 415: Submarine gate

Cashew gate

A cashew or banana gate is a curved tunnel gate. The mesh for a cashew gate is shown in Figure 416. A cashew gate is used to put the gate orifice on the bottom edge of the part where it will be hidden. The cashew gate is more difficult to manufacture, as it has to be made as a two-piece insert. If a piece of polymer gets stuck in the gate, it can often be difficult to remove without removing the gate insert out of the mold. Stiffer polymers cannot be used with this type of gate, as the polymer is significantly deformed during ejection. The part is de-gated with a tensile break. To prevent major problems with de-gating, the orifice has to be small. This leads to higher shear rates.

Midplane and Fusion meshes

There should be at minimum 3 elements defining the sub gate, preferably more. Since the gate is tapered, a better freeze time prediction will be achieved with more elements.

3D meshes

Cashew gates can easily be modeled with beam elements for a part that has a 3D mesh. Tetrahedral elements can be used and with enough mesh density the shear rate and pressure drop will be more accurate. Meshing round features tends to take many elements. You must make sure there are still several rows of elements down the length of the gate.

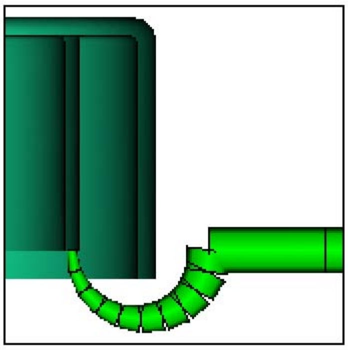


Figure 416: Cashew gate

Pin gate

A pin gate is a gate used in a 3-plate tool. The runners are on a secondary parting line. A cold drop goes through the runner plate. At the end of the drop, there is the actual gate. Figure 417 shows a pin gate and it's mesh. The gate has a very small diameter and a large included angle. Normally it is blended into the end of the drop. This gate breaks with a tensile force much like a cashew gate. The orifice must be small to keep the gate mark small as well. The shear rate is always way too high, but is limited by the need for a clean break and small witness mark.

Midplane and Fusion meshes

The gate is modeled with tapered circular beam elements. There should be at least 3 elements for the gate, and several more for the drop.

3D Meshes

Pin gates can easily be modeled with beam elements for a part that has a 3D mesh. Tetrahedral elements can be used and with enough mesh density the shear rate and pressure drop will be more accurate. Meshing round features tends to take many elements. You must make sure there are still several rows of elements down the length of the gate.

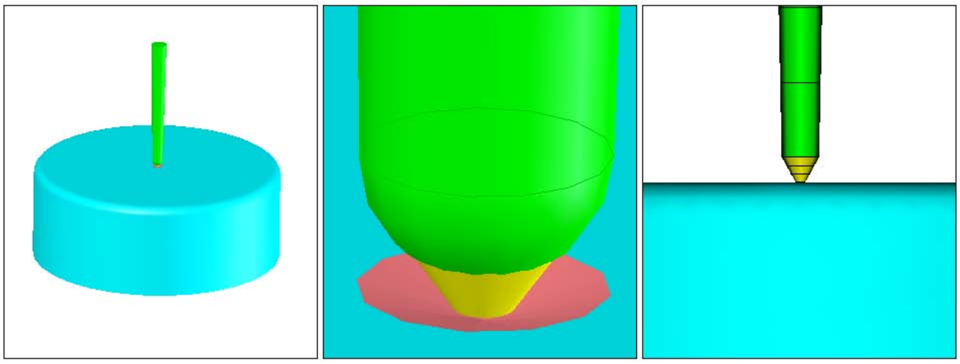


Figure 417: Pin gate

Hot drop

A hot drop is a portion of a hot runner system that delivers the polymer from the hot manifold system to the part. The gate orifice itself can be many different configurations, but is generally round or annular. Figure 418 shows a generic hot drop configuration and the mesh of the gate orifice. The gate orifice can be straight or tapered. The actual geometry will need to be obtained by the hot drop vendor.

Midplane and Fusion meshes

Beam elements are used to model the hot drop. It is important to model the orifice with at least 3 elements if the prediction of gate freeze is important. With a hot gate property, you have the ability to set the outer heater temperature. The default is the melt temperature. This is generally good. However, the gate tip is probably somewhat below the melt temperature. It is probably closer to the transition, or ejection temperature of the material. The temperature can be set if desired.

3D meshes

A hot drop can be modeled with beam elements or tetrahedral elements. If tetrahedral elements are used, the property must be set to **Hot runner (3D).**

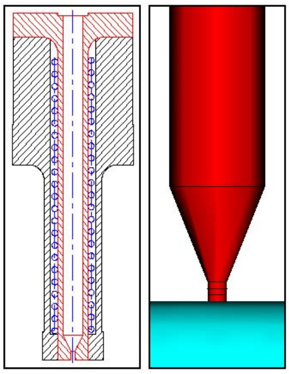


Figure 418: Hot Drop

Valve gate

A valve gate is a special version of hot drop where there is a controlled pin that creates a positive shutoff as shown in Figure 419. The flow channel through the drop is generally annular but can be circular.

Midplane and Fusion meshes

To define a valve gate, a valve gate controller is associated with the last element before the part. This controller can then have several methods to control the valve gate. Every valve gate that requires a different control must have a different controller assigned to it.

3D meshes

To use valve gates on a 3D tetrahedral mesh, the valve gate needs to be created with beam elements just like Midplane and Fusion models.

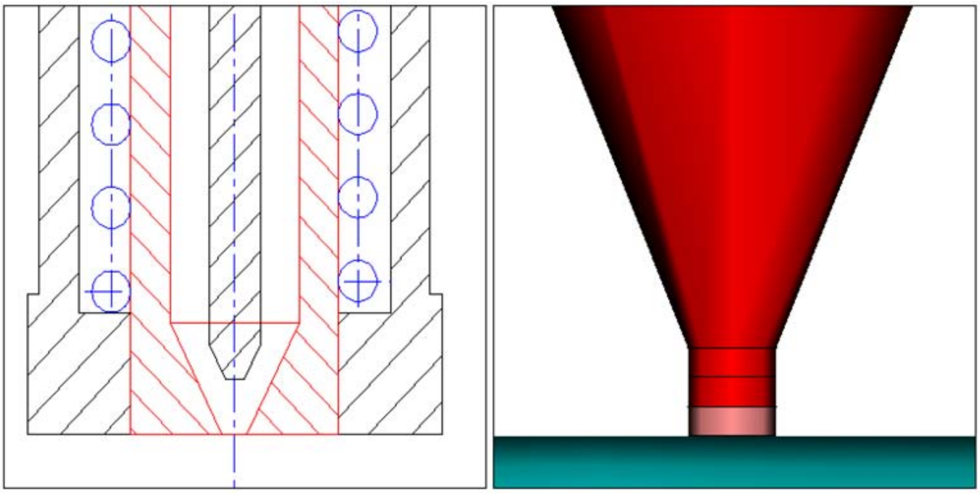


Figure 419: Valve gate

Gate sizing

When sizing the gate, several things should be considered. These include:

- The shear rate.
- Nominal wall of the part.
- The type of gate.

From a flow analysis point of view, the shear rate is the most important. Every material in the database has a **Maximum Shear Rate limit** on the **Recommended Processing** tab for the material. Generally shear rate limits range from 30,000 to 60,000 1/sec. Polypropylene has a shear rate limit of 100,000 1/sec. It is generally desirable to keep the gate shear rate below, or even well below the shear rate limit.

Additives are more sensitive to the high shear than the polymer itself, so if there are additives the shear rate is more important. This can include fillers such as glass, or talc, stabilizers, or colorants for example. Targeting a very low shear rate, of **20,000 1/sec.** is desirable when possible.

Depending on the type of gate, the size may or may not be increased to get the shear rate down. Edge gates are generally easy to get the shear rate down. Increase the thickness, and then if the shear rate is still too high increase the width. Sub gates, and hot drop orifices are more difficult to gate in order to create low shear rate; open the gate as large as it is practical.

Setting the Gate Cross-Section

When creating curves or beams the property of the new entity can be set using the

Create as field. If the property listed in the field is not correct, Click the  icon. The assign property dialog will appear as shown in Figure 420. The **Select** button accesses databases for the entity types, and the **New** button allows the user to set all the specific properties as necessary.

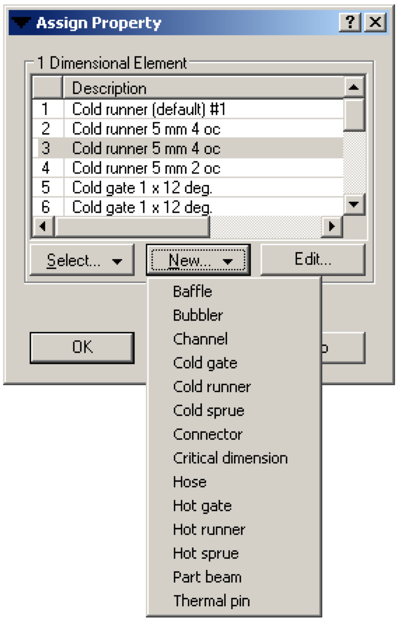


Figure 420: Assign properties to beams

Once a gate type has been selected, the specific properties for the gate can be entered. Figure 421 shows the properties for a hot gate. They include:

- The cross section.
- The shape.
    - Non-tapered.
    - Tapered by angle, shown in Figure 422.
    - Tapered by end dimensions.
- Sizes.
- Heater temperature.
- Heat loss to into the mold.
- Occurrence number.
- Valve control.
- Mold properties.

The heater temperature, heat loss to into the mold, and valve control are unique to hot gates.

The taper angle, is measured from the beam centerline to the edge and is half the included angle.

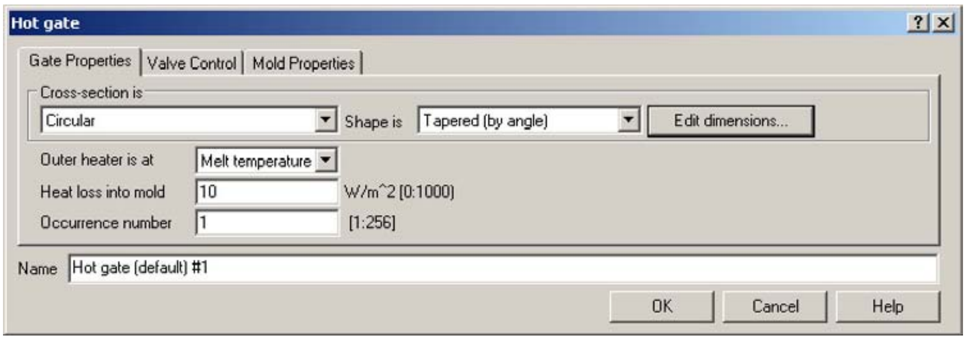


Figure 421: Hot gate properties

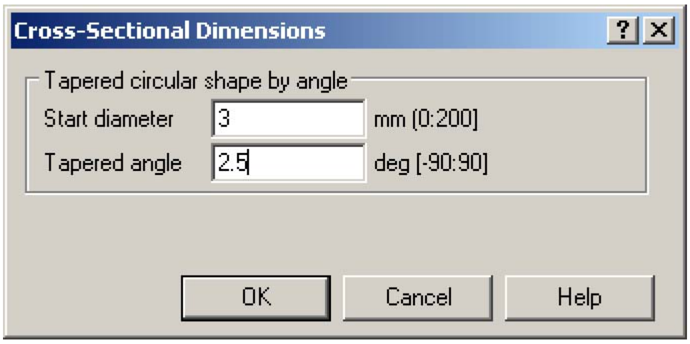


Figure 422: Cross-sectional dimensions

Runner design

When designing runners, there are 3 main attributes that need to be considered, the runner layout, the cross-sectional shape, and the runner size.

Runner layouts

There are many types of runner layouts, but there are 3 main categories of runner layouts as discussed below.

Herringbone (Standard)

With the herringbone layout, there are two rows of cavities. The sprue is in the middle, and the number of cavities is generally a multiple of four. This runner system is not naturally balanced, as the flow length between the sprue and the parts is not the same for all the cavities. In Figure 423, there are two groups of cavities, the inner and the outer 4 cavities. These two groups of cavities will not fill at the same time unless the runners are balanced. The runners can be balanced by changing the runner diameters. The secondary runners to the inside cavities would be made smaller than the outside cavities. The runner balance analysis can be used to size the runners.

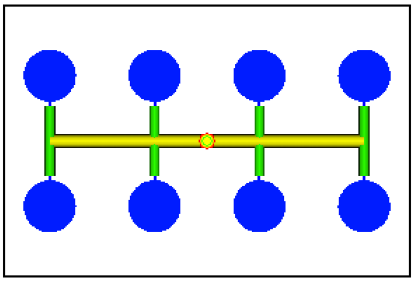


Figure 423: Herringbone runner layout

Geometrically balanced (“H” Pattern)

A geometrically balanced runner, traditionally called naturally balanced, shown in Figure 424, has the same flow length between the sprue and each part. The number of cavities is a power of two. This type of runner system is considered the best because it will have a wider processing window than a herringbone runner system, even if the herringbone system is balanced. The disadvantage to this type of runner system is a higher runner volume, and more room needed between the rows of cavities to allow room for the runners.

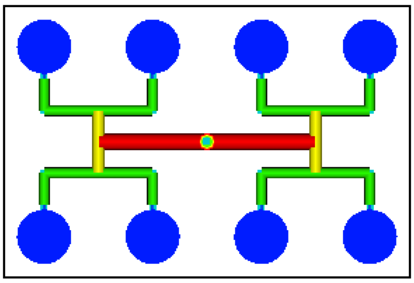


Figure 424: Geometrically or naturally balanced runners

Radial

With a radial layout shown in Figure 425, the cavities are put in a circle around the sprue, and a runner goes between the sprue and the cavities. This is also a naturally balanced runner layout.

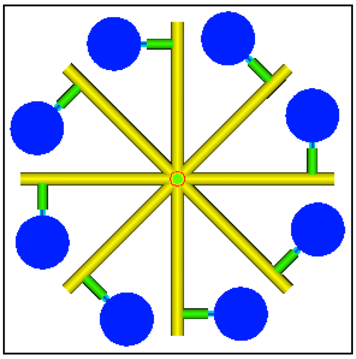


Figure 425: Radial runners

Runner cross-sectional-shapes

Runners can be cut in several different cross sectional shapes, shown in Figure 426. A full round, or circular, is the best shape to use. It is also the most expensive as it needs to be cut on both sides of the parting line. When the parting line of the mold is not flat, generally some other shape is used. Trapezoidal, or modified trapezoidal - sometimes called "U" shaped, are good substitutes for the circular shape.

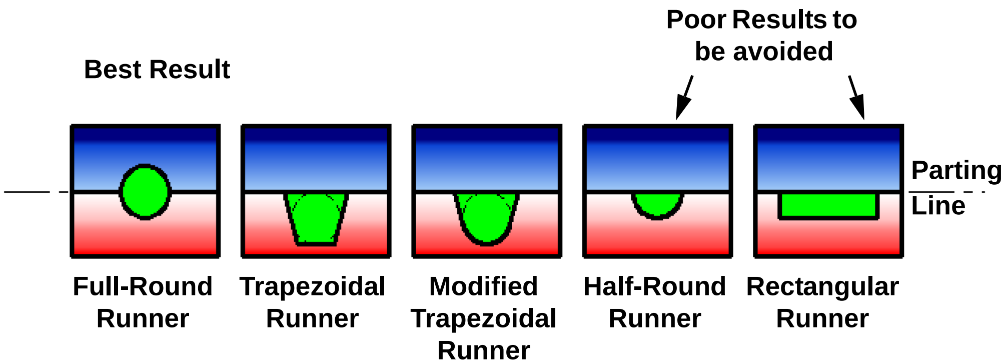


Figure 426: Runner Cross-Sectional Shapes

The shape of a trapezoidal or a "U" shaped runner should be like Figure 427. The depth of the runner should be equal to the diameter of a circular runner and the sides should be tangent to the diameter of the runner.

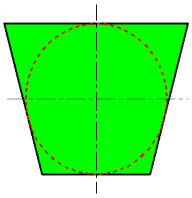


Figure 427: Circular vs. trapezoidal runner

Runner sizing

The size of the runners is dependant on many factors including, the material, flow length, and pressure requirements of the part being filled. Table 70 lists typical runner diameters for different materials. The range is very wide to account for typical flow lengths, pressure drops and material viscosity variation within a material type. The sizes are a good starting point. In general, the smallest possible runner size is desirable to minimize the material usage. The Flow analysis will help determine if the size runner being used will allow the part to fill and pack adequately.

Table 70: Typical runner diameters for unfilled generic materials

	Diameter			Diameter	
Material	mm	inch	Material	mm	inch
ABS, SAN	5.0-10.0 3	3/16-3/8	Polycarbonate	5.0-10.0	3/16-3/8
Acetal	3.0-10.0	1/8-3/8	Thermoplastic polyester (unreinforced	3.0-8.0	1/8-5/16
Acetate	5.0-11.0	3/16-7/16	Thermoplastic polyester (reinforced)	5.0-10.0	3/16-3/8
Acrylic	8.0-10.0	5/16-3/8	Polyethylene	2.0-10.0	1/16-3/8
Butyrate	8.0-10.0	5/16-3/8	Polyethylene	2.0-10.0	1/16-3/8
Fluorocarbon	5.0-10.0	3/16-3/8	Polyphenylene oxide	6.0-10.0	1/4-3/8
Impact acrylic	8.0-13.0	5/16-1/2	Polypropylene	5.0-10.0	3/16-3/8
Nylon	2.0-10.0	1/16-3/8	Polysulfone	6.0-10.0	1/4-3/8
Phenylene	6.0-10.0	1/4-3/8	Polyvinyl (plasticized)	3.0-10.0	1/8-3/8
Phenylene sulfide	6.0-13.0	1/4-1/2	PVC Rigid	6.0-16.0	1/4-5/8
Polyallomer	5.0-10.0	/16-3/8	Polyurethane	6.0-8.0	1/4-5/16

Branched runners

In geometrically balanced runner systems, it is common for the runners to reduce in size from the sprue to the gates. The change in size would occur when the runners split or branch. Figure 428 shows an example of changing the runner size. It is best to calculate the runner sizes using the constant pressure gradient principle using Moldflow runner balancing analysis. However, the sizes can be approximated using the following formula:

$$d_{feed} = d_{branch} \times N^{1/3}$$

Where:

d_{feed} = the diameter of the runner feeding the branch.

d_{branch} = the diameter of the runner branch.

N = the number of branches.

In a geometrically balanced runner system, the number of branches will always be two.

For the model in Figure 428, the diameter of the runner, at the gate is 4.0 mm and is the starting point for the calculations. The number of cavities is 8, so there are 2 branches in the runner system. To calculate the secondary runner:

$$5.04\text{ mm} = 4\text{ mm} \times 2^{1/3}$$

The secondary runner diameter was rounded to 5.0 mm. The primary runner is calculated based on the secondary runners and is 6.35 mm.

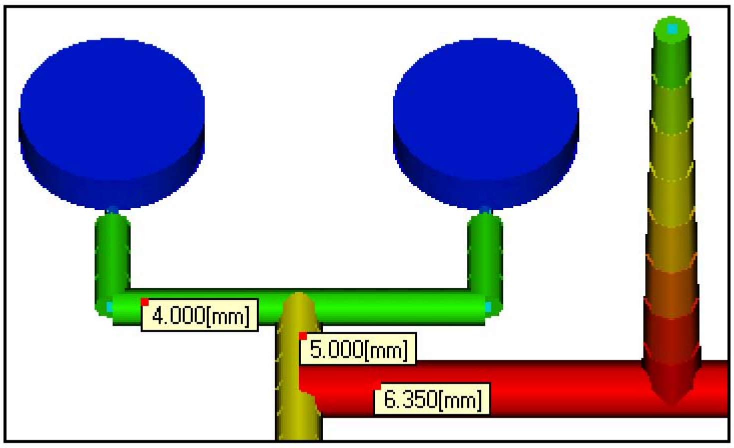


Figure 428: Runner diameters calculated based on branching

Runner creation

When creating a multi cavity tool there are two wizards that can be used to assist in creating multiple cavities and creating the runner system. There are also two different techniques for manually creating runners.

Cavity Duplication Wizard

The cavity duplication wizard, shown in Figure 429, is a tool to help quickly duplicate the part for a multiple cavity tool. To use the wizard enter:

- The number of cavities.
- The number of columns, or rows.
- The column, and row spacing.
- The spacing is center-to-center spacing.

The check box **Offset cavities to align gates** is a feature that can be used to align the gates rather than the cavities. For example, if the gate should be on the centerline of the part but is not exactly on the centerline, if this check box was not checked the parts would be aligned, and the runners would be at a slight angle.

The preview button should be clicked to see what the layout would look like. The red cavity is the original. There is also a yellow dot. This is the gate location. It may be necessary to cancel this wizard and manually orient the original part so it is the same as the red original cavity. The cavity duplication wizard assumes that the **parting plane is the XY plane**.

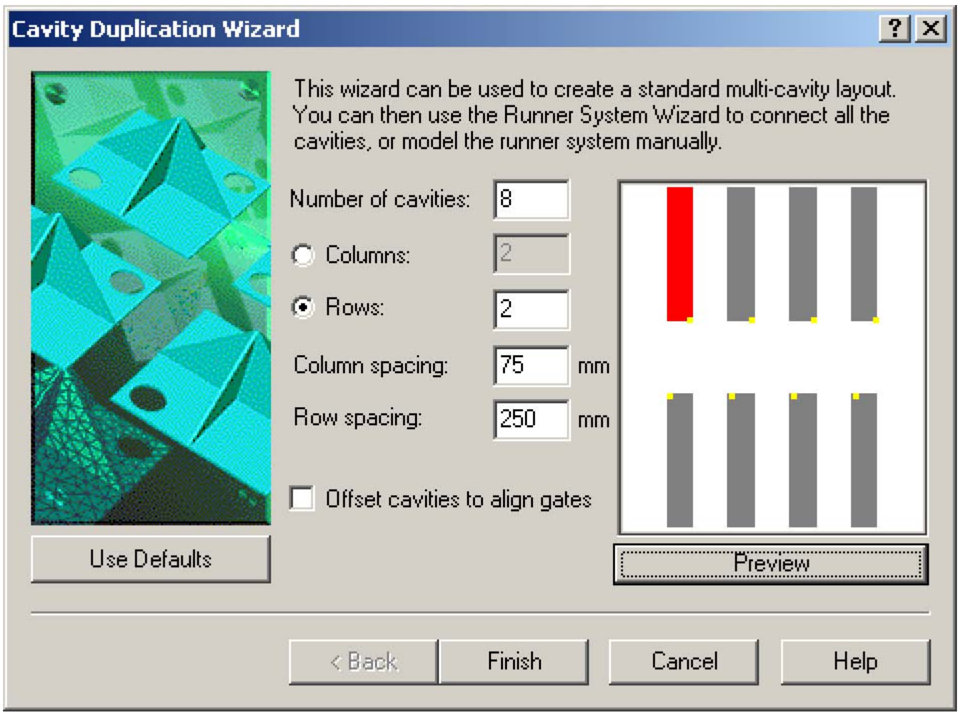


Figure 429: Cavity duplication wizard

Runner System Wizard

The runner system wizard is designed to create a **geometrically balanced runner system** with tunnel gates. If edge gates needed, this wizard will still work, but the gate geometry needs be set after the wizard is run. It assumes the parting plane is the XY plane and the positive Z direction is toward the nozzle of the machine. The completed runner system is placed on the active layer. You may want to create a new layer and make it active before running the wizard. When the wizard is executed, it looks at the model and sets up some defaults.

Page 1

The first page of the wizard is shown in Figure 429. The inputs on page one are described below.

Sprue location

The location of the sprue defaults in the center of the mold. The sprue location is set by clicking the buttons, Center of mold, Center of gates or entering in the X and Y coordinates. The location defined by the Center of mold may be different than the center of gates, depending on the cavity layout and gate positions.

Gate types

The runner system wizard looks at the injection location marker (yellow cone) and determines if the gate is a **side gate** or a **top gate**. A side gate would be an edge, or sub gate and the injection location cone is in the XY plane. A top gate represents a gate for a 3-plate tool or a hot drop. The injection cone for a top gate would have its axis in the Z direction. Fields that are available depend on the types of gates found.

Runner type

A check box is used to determine if hot runner or cold runners are used. Depending the type of gates and runners used, the tool will assume one of four options:

- If the gates are side gates, and the runners are cold, it would use a cold sprue and cold runners.
- If the gates are side gates and the runners are hot, there will be a hot manifold and drop, plus cold runners and gates.
- If the gates are top gates and the runners are cold, it will create a 3-plate type layout.
- If the gates are top gates, and the runners are hot, all the runners will be hot and naturally balanced. The gates will be hot.

Gate location

The parting plane Z field defines the Z coordinate the runners will be created in. Three buttons can be used, top, bottom, and gate plane to define the parting plane. The gate plane button can be used if the gates are edge gates. The top and bottom will be the highest and the lowest Z dimensions for a bounding box around the part. If the parting plane is not at any of these locations, you must enter the Z value.

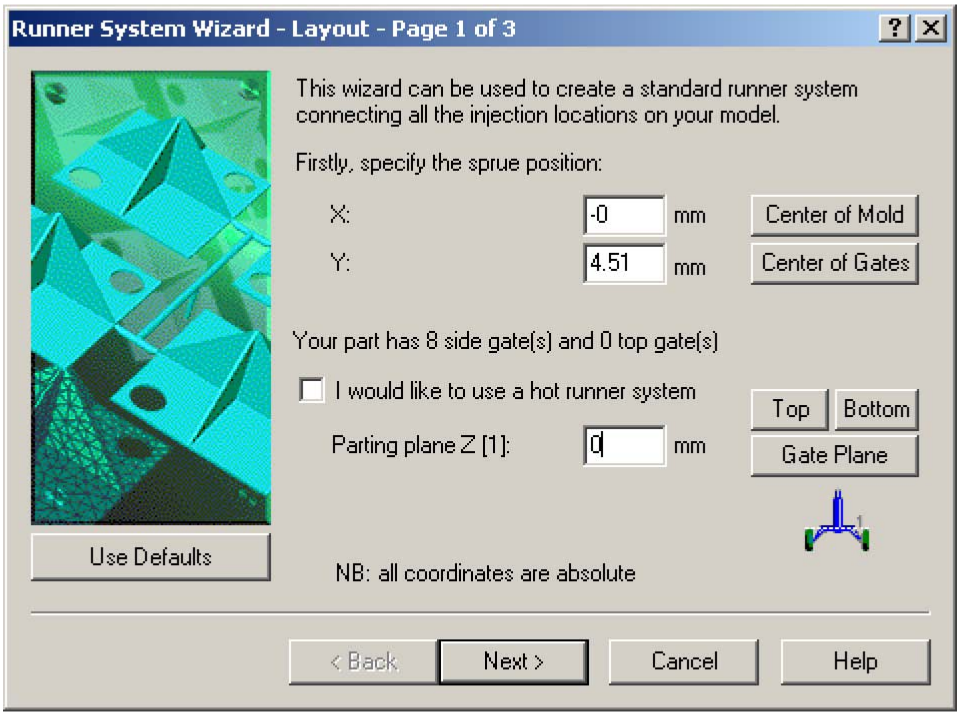


Figure 430: Runner system wizard, page one

Page 2

Figure 431 shows an example of the second page of the wizard. On this page you set the sprue, runner and drop information as needed. If the fields are grayed out, the information is not needed.

Sprue

The sprue, needs the following set:

- Orifice diameter.
- Included angle.
- Flow length.

For a standard DME sprue, the taper is ½ inch per foot, which equals the included angle of 2.38 °. If you were to enter the sprue angle manually in the sprue properties dialog, the taper angle would be 1.19°.

Runners

The runners can be entered as round, or trapezoidal. When the runners are trapezoidal, the included angle is also required.

Drops

When drops are defined, a bottom diameter and included angle are required.

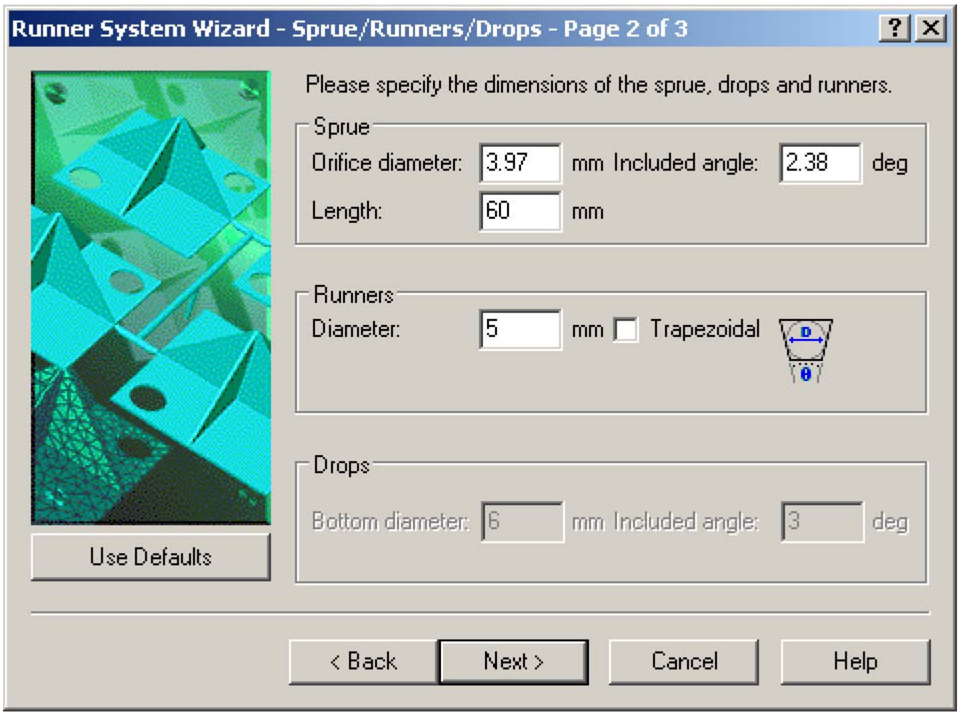


Figure 431: Runner system wizard, page two

Page 3

Gate information is entered on the third page of the wizard, shown in Figure 432.

Side gates

The wizard assumes side gates are tunnel gates so the cross-section is round. Information required for a side gate includes:

- Orifice diameter.
- Iincluded angle.
- Length or angle to mold face.

If the gate is really an edge gate, enter any value for the orifice diameter, **0** for the included angle, and enter the land length for the gate. After the wizard has built the runners, click on a gate element. Edit its properties to set the cross-section to rectangular and the correct cross-sectional dimensions. Since the included angle is zero, all elements will have the same property. The wizard will force at least 3 elements in the land length of the gate.

When the side gate is a tunnel gate, then the angle to mold face is normally easier to enter than the length.

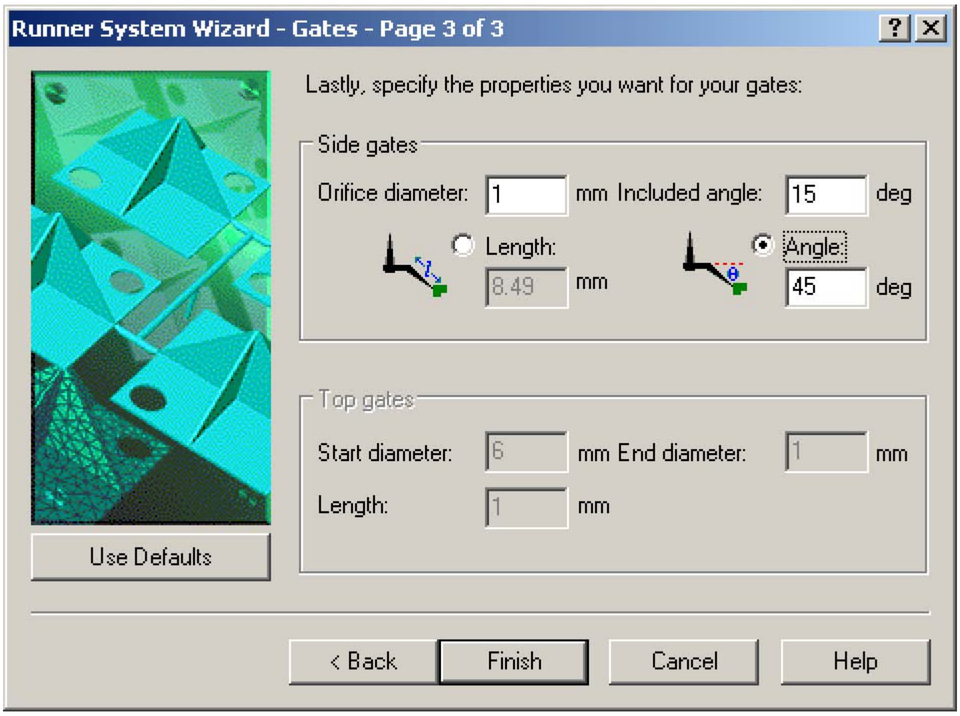


Figure 432: Runner system wizard, page three

Manual runner system creation

Normally, the feed system is constructed after the part has been analyzed and optimized. The order in which you create a multi-cavity model from a single cavity depends to some degree on the tool layout and how you want to approach modeling the geometry. When you have a multi-cavity tool, the cavities can be constructed in two ways, the Cavity Duplication Wizard, and manually copying the cavities. Many times if the cavities are manually copied, it is done in conjunction with runner creation.

Tool position

You may want to consider the position of the cavities. You will find it easiest to model the runner system and later the cooling system if the part is in **Tool Position**. There are two levels of tool position. The first defines the parting plane and the direction of the sprue. In MPI, the **XY plane** is considered the parting plane. The following tools or calculations assume the XY plane is the parting plane:

- Cavity Duplication Wizard.
- Runner System Wizard.
- Cooling Circuit Wizard.
- Clamp force calculations in a fill or flow analysis.

The second level of tool position is the location of the origin, 0, 0, 0. In most tool designs, the origin is the center of the mold base at the main parting line. The sprue in most cases is in the center of the tool with the bottom of the sprue at the parting line. Therefore, the base of the sprue is the origin. You will find it easy to model feed system and cooling system components if the part model is in tool position both regards to the XY plane, and the origin of the model.

Moving geometry to tool position

With the tool **File ➡ Move/Copy**, you have several tools to help you move your part(s) to tool position. Figure 433 shows the **3 Points Rotate** tool. This tool will allow you to re-define the global coordinate system for your part. To use the tool do the following:

1. Select the entities you want to move. Typically, it will be everything in your model on all layers.
2. Select **Coordinate 1**. This is the new origin. Typically, it is a node on your part. Set the filter to node to ensure you have an exact node location. Once the XY plane is defined correctly, the origin can be moved to the base of the sprue location using a different tool.
3. Select **Coordinate 2**. This defines the new global X axis direction. Normally a node on the part is used.
4. Select **Coordinate 3**. This will define the XY plane and using the right-hand rule the direction of the positive Z direction. As with coordinate 1 and 2, a node on the part is normally used.

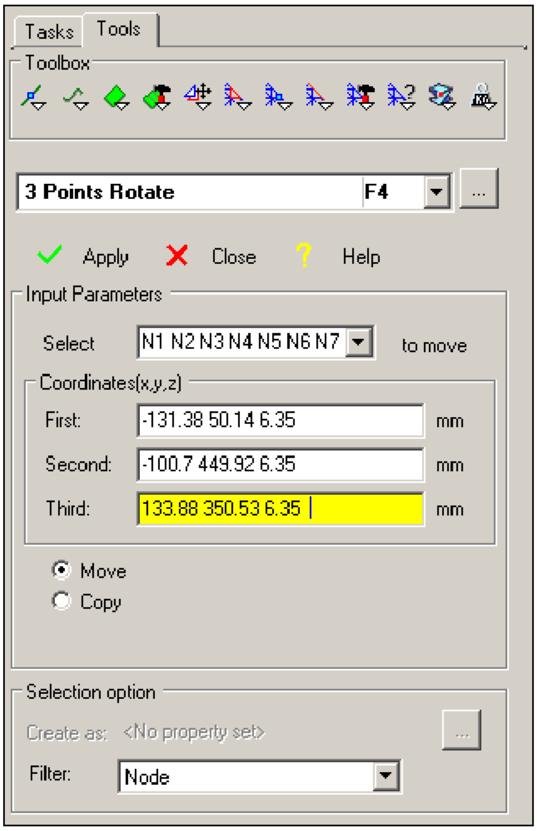


Figure 433: 3 Points rotate

Once the XY plane is properly defined, the origin can be positioned in the correct location, relative to the existing geometry in your study. Use the measure tool to determine the current location of some key node on your model. Calculate how far off in X, Y, and Z it is. Use the Translate tool shown in Figure 434 to move all the geometry in your study to the correct location.

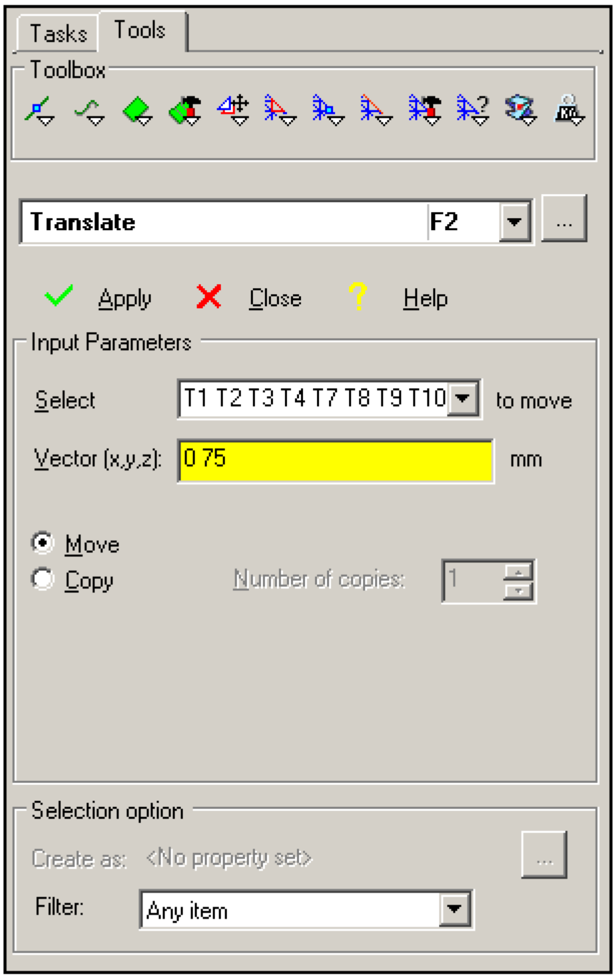


Figure 434: Translate

Entity properties

When modeling runner systems, there are some entity properties that you will need to set or make sure they are correct including:

- Occurrence number.
- Cross-sectional shape and size.
- Runner Constraints.

Occurrence number

Occurrence numbers account for symmetry within a flow path. In Figure 435, the tool is geometrically balanced. This means that the flow length from the sprue to the end of each cavity is the same. There is only one unique flow path in the part. When using occurrence numbers, the only geometry that must be modeled is the sprue, the runners leading to one of the cavities, a gate, and the cavity. Occurrence numbers account for the runners, gates and parts that are not physically modeled, significantly reducing the number of elements needed to represent the geometry. This is particularly useful for runner balancing because it is an iterative process and reducing the number of elements will speed up the analysis significantly.

In Figure 435, the flow path with the solid lines from the sprue to the upper left cavity can represent the entire tool. The numbers along the flow path represent the occurrence numbers for that portion of the flow path. The sprue occurs once, the main runner occurs twice, the secondary occurs four times, and the remainder of the runner system, gates and parts occur eight times. The dashed lines are runners, gates and cavities that are represented by the occurrence numbers.

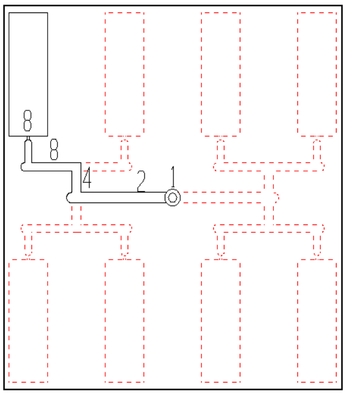


Figure 435: Occurrence numbers of a geometrically balanced feed system

In Figure 436, the eight-cavity herringbone runner system can be represented with occurrence numbers as well. In this example, there are two distinct flow paths that represent the tool. One flow path goes to the outer four cavities and another flow path for inner four cavities. In this case, the sprue and main runners still have the occurrence number of 2 as in the first example. In this case, the secondary runners, gates, and parts all have an occurrence of four.

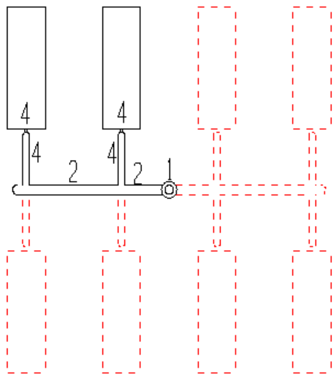


Figure 436: Occurrence numbers of a herringbone feed system

Cross-sectional shape and size

When setting the cross-sectional shape and size for feed system entities, make sure they are correct. In many cases the defaults are correct, but you should always check to make sure they are correct.

Runner constraints

When building a model for runner balancing, you need to consider the constraints on the runner. During runner balancing, beam entities that are gate, or sprues are not changed, but runners by default are. In the Cross-Sectional Dimensions dialog for a cold runner, you can edit the runner constraints as shown in Figure 437. The constraint choices are:

- Fixed.
  - The runners will not be changed at all during the runner balance.
- Constrained.
  - The runners can change within user defined tolerances. This may limit the amount of balancing done and my not allow the best possible balance.
- Unconstrained.
  - The runners can change to any size needed to achieve a balance. This is the default.

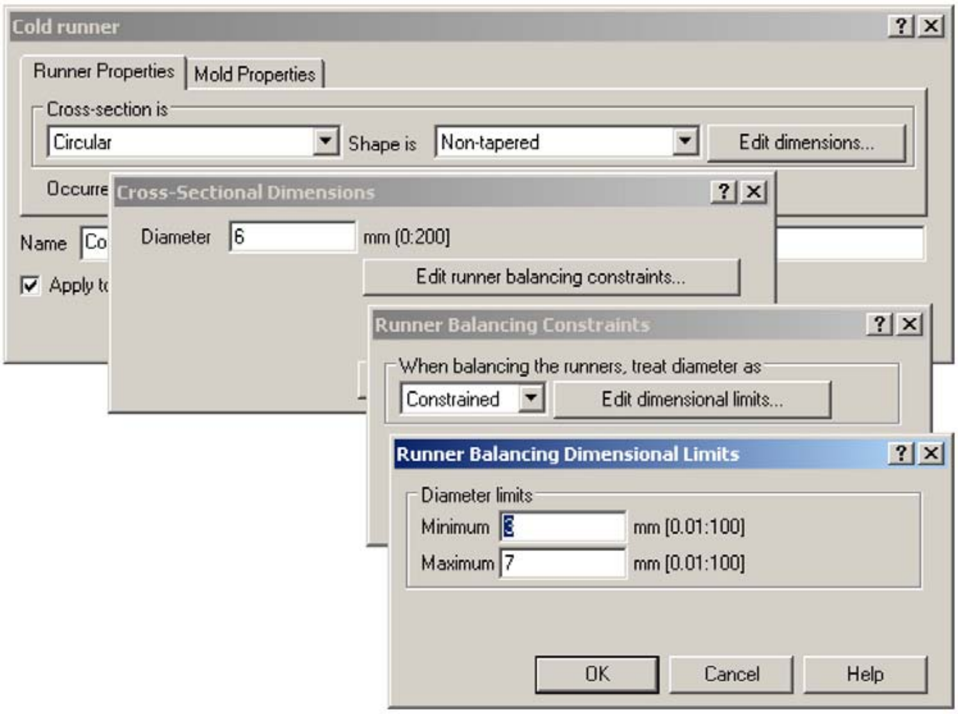


Figure 437: Runner constraints

In most cases, leaving the constraint setting to the default value of Unconstrained is best, at least for the first iteration of the balance.

Feed system construction

When creating runner systems manually, there are two basic approaches for creating beam geometry:

- Create curves then mesh the curves.
- Create the beams directly.

Both methods have advantages. Many times it is a personal preference. The methods will be discussed below.

Create curves then mesh the curves

When you create curves, you define the center line of the curve by the end coordinates, and the cross sectional shape. An example of this is shown in Figure 438. The advantage is that you create the entire feed system at one time, or at least you can. This ensures a consistent mesh density. One disadvantage is you may want different mesh densities. In particular, you want to make sure the gate has at least 3 rows of elements in the gate. You would not want that fine mesh density on the runners; you would have a mesh that is too fine. What you should do is create the curves for the gate(s) and mesh the gates at the appropriate density, then create the curves. You can use layers to assist you. If the gate curves are on a different layer than the runner curves, you can make the curves at the same time, just mesh one layer at a time. The mesh generator will only mesh entities on visible layers, so turn off the layers you don't want meshed.

When creating curves, it is best to set the property of the curve before creating it. You can assign properties afterwards, but it is easiest if you do it before making the curve.

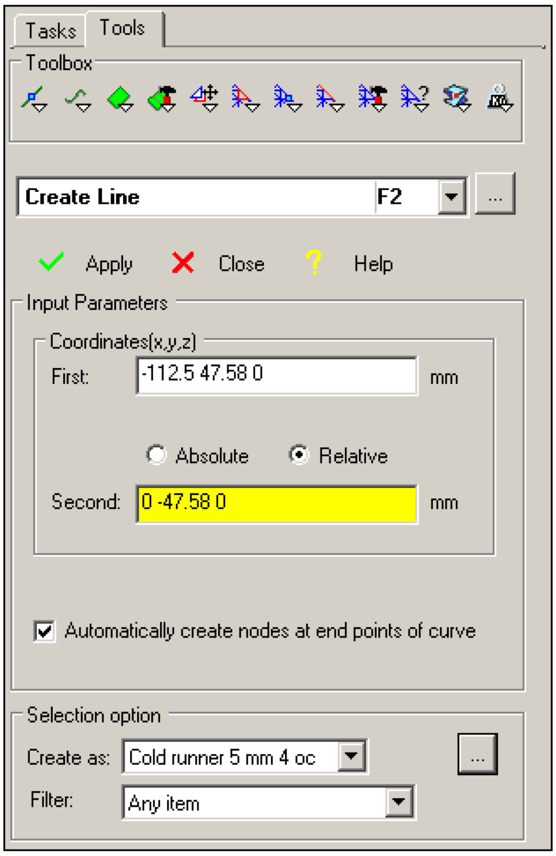


Figure 438: Create curves

Create the beams directly

If you create beams directly using the dialog shown in Figure 439, you need to define the endpoints then the number of elements to create. In most situations, it will take some time to determine the correct number of elements to create. When the elements are created, the underlying curve is also created. This tool works well for uniform cross section geometry such as runners. However, it does not work as well for sprues or tunnel gates. Every element created between the coordinates entered will have the same cross-sectional size. If a sprue was constructed with 6 elements, by this method, every element would have the same start and end diameter. If curves were constructed then meshed, then the end element diameter becomes the start element diameter for the next element.

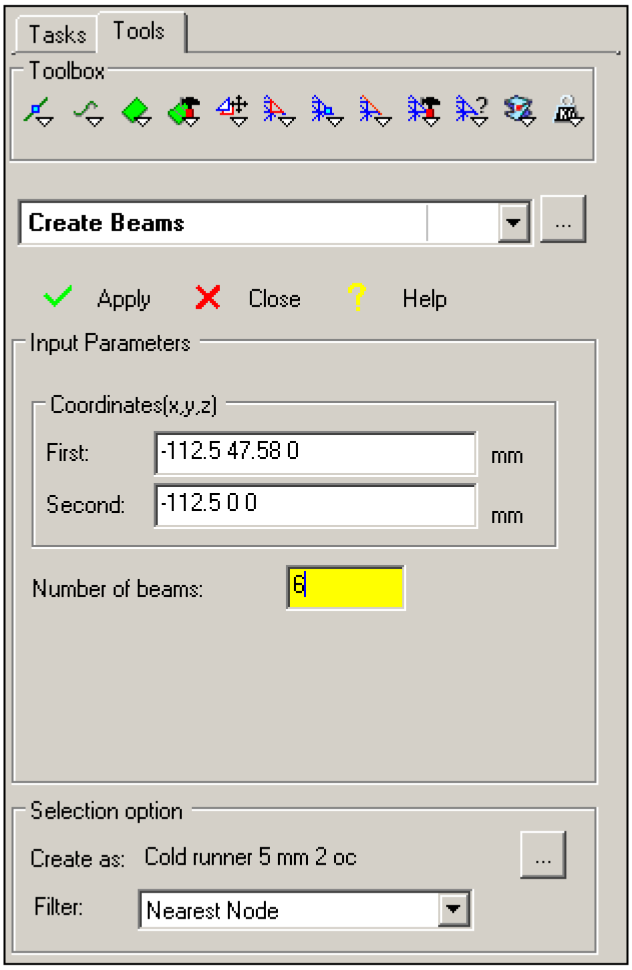


Figure 439: Create beams

Length to diameter ratio

The **L/D ratio** for beam elements should be about **2.5:1** to **3.0:1**. This is mostly for cooling analysis work not flow. Since the same study is used for both, the beam elements for the fill should be constructed so they don't cause a problem with cooling.

It is not possible to achieve this ratio in gates and still achieve the 2.5:1 L/D ratio. In this case, the need for 3 elements in the gate out weights the requirement for the ratio in beam elements.

Runner balancing

Runner Balance is an analysis sequence. This sequence includes a filling analysis then the runner balance. During the runner balance, the runner sizes are changed so each flow path takes the same time to fill and the same amount of pressure within defined tolerances. A target pressure set in the Process Settings Wizard controls the balance process.

A target pressure set in the Process Settings Wizard controls the balance process.

Why Balance Runners

Runner systems should be balanced so that all the cavities in the tool fill and pack at the same time and pressure. Many times when parts do not fill in the same time, one part may have flash or burn marks while other cavities in the tool do not. When tools are balanced, there is less variation between the cavities and there is a larger processing window. Runners are sized by the runner balancing process. The size of the runners are determined by the target pressure used in the analysis. Even a geometrically balanced runner should be sized. The runner volume should be as small as possible without causing packing problems.

Runner Balancing Procedure

Runner balancing is an iterative process but there are some basic steps that must be followed to achieve a good runner balance including:

1. Optimize the part.
2. Model the runner system.
3. Run a fill analysis with the runner system.
4. Determine the target pressure.
5. Run the balance analysis.
6. Review the results.
7. Revise the target pressure as necessary and re-run.
8. Round the runner sizes if desired.

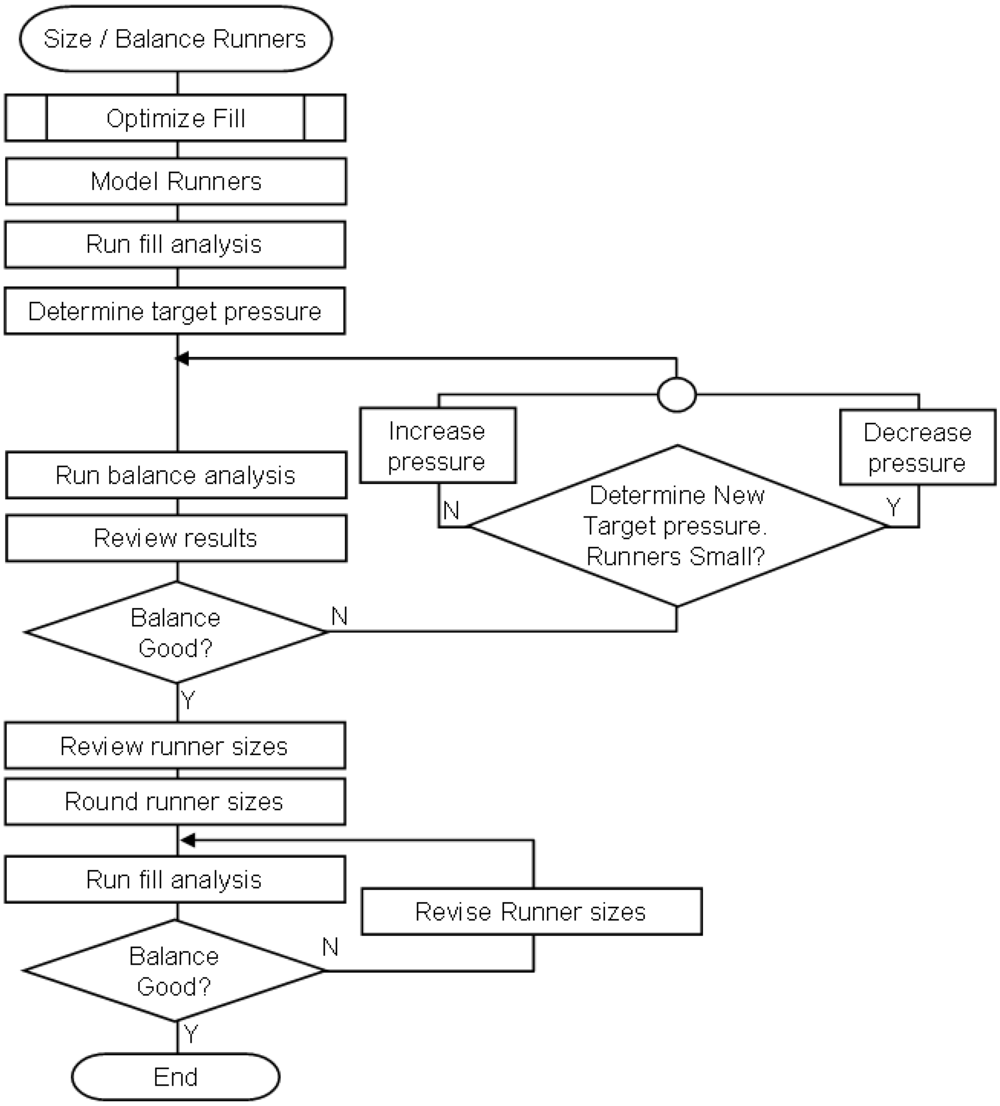


Figure 440: Runner balance flow chart

Optimize the part

All optimization work on the part must be done before runner balancing is to begin. In particular, the molding conditions and gate location must be set.

Model the runner system

The runners system must be created using the techniques described above. The runners that are not to be sized must have their constraints set to **Fixed**.

Run a fill analysis with the runner system

A filling analysis is done on the study that contains the feed system and the parts to determine the amount of imbalance in the tool and the target pressure for the balance. To run the filling analysis, the processing conditions below must be properly set:

- Set the analysis sequence **Fill**.
- Change the following process settings.
  - Flow rate.
  - Velocity to pressure switch-over.
  - Packing pressure.

The fill control needs to be changed to flow rate, so the injection time is required to fill the cavities is achieved even with the increase in runner volume. To determine the flow rate:

$$\text{Flow Rate} = \frac{\text{Total volume of the parts}}{\text{Injection time for the part}}$$

The volume of the parts does not include the runner system. The injection time is the optimized injection time based on previous work with injection locations on the part.

The switch-over and packing pressure need to change from their default values. In general, it is good practice to change the switch-over to 100% volume, and the packing pressure to 100% filling pressure. After the fill analysis is complete and results interpreted, the analysis sequence of this study is changed from fill to runner balance. The runner balance will start based on the results of the completed fill analysis. The runner balancing analysis works better when the pressure does not drop after the switch-over. Once the runners are properly balanced, the switch-over can be adjusted to the desired values.

Determine the target pressure

The target pressure is the key input for a runner balance analysis. The runner balance will increase or decrease the size of runners not only to create a balance, but to ensure the pressure required to fill the tool is within the tolerance centered around the target pressure. For instance, if the target pressure is 50 MPa, and the default pressure tolerance of 5 MPa were used, when the fill pressure is between 45 and 55 MPa the pressure tolerance has been met. If the target pressure is too low, the runner diameters will be large. When the target pressure is too high, the runners will become too small.

Determining the target pressure is an educated guess at best. This is particularly true when you want to optimize the runner volume by having the smallest runners that will still properly fill and pack the parts.

View the **Pressure at injection location XY plot**, shown in Figure 441 as a guide for setting the target pressure. The fill pressure is about 70 MPa. The target pressure should normally be similar to the pressure to fill, If there is a significant pressure spike at the end of fill the target pressure should be a bit lower than the fill pressure. In Figure 441, the pressure spike is not great, so a good balance pressure to start with is approximately the fill pressure of 70 MPa.

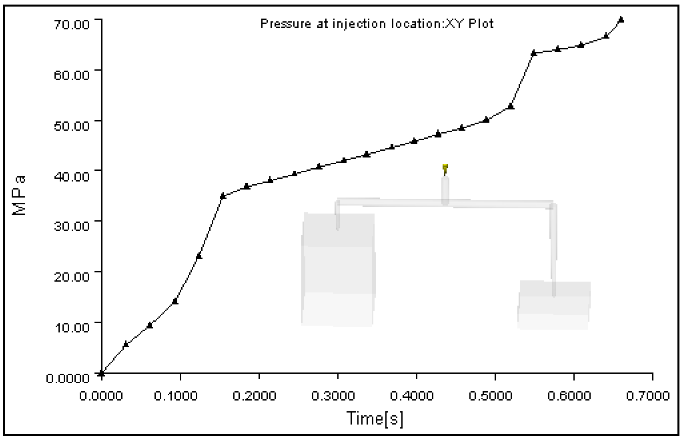


Figure 441: Pressure at the injection location

Run the balance analysis

The runner balance is started by setting the **Analysis sequence** to **Runner balance**, setting the process parameters necessary and launching the analysis. In addition to the target pressure discussed above the runner balance has some advanced options shown in Figure 442. These options can be adjusted to help minimize the imbalances between cavities. Generally the tighter you make the tolerances the better the balance but also longer the analysis time will be. Most of the time, the options are not changed for the first runner balance analysis. If additional analysis is necessary, the options may be changed.

Mill tolerance

The mill tolerance defines the increment in size the runners will take at each iteration of the runner balance analysis. The default value is 0.01 mm. If the tolerance was opened to 0.1 mm, at each runner balance iteration, the runners diameters will change 0.1 mm. If the tolerance is opened too far, the quality of the balance will be affected. It will also limit the number of iterations it can do before no more changes can be done.

Maximum Iterations

This limits the number of iterations the analysis will do. Each iteration is a fill analysis. The runner balance will stop if it reached the iteration limit set or it meets the tolerances set. When a model is large, the runner balance can take a very long time if many iterations are required.

Time convergence tolerance

This is the difference in fill time between the first cavity filled and the last cavity filled expressed as a percentage. The smaller the percentage, the better the balance. The default is 5%. This value may be too high for smaller parts.

Pressure convergence tolerance

The pressure tolerance is the difference in pressure between the fill pressure and the target pressure. The default is 5 MPa (725.2 psi). This tolerance will influence the size of the runners, and the balance. The size is influenced because the size is determined by the target pressure. The tighter the pressure tolerance, the more control you will have on the runner diameter. It will take more iterations to meet the tolerance as well, so the analysis will take longer.

Figure 442: Runner balance settings

Review the results

When a runner balance analysis is completed, there are two studies to review. The first study is the one you launched the runner balance analysis on. It will have the icon  listed by it in the project view. This indicates it is a fill analysis with an optimization analysis. The second study contains the revised runner sizes and is a fill analysis. This study is the main one used to determine the acceptability of the balance. The name of the study is the name of the first study with **(Runner Balance)** appended to its name.

First study results

This first study will have two results to review, the screen output and a volume changed plot. Figure 443 is an example of the iteration table for a runner balance from the screen output. Each iteration is a flow analysis. It shows how well the runner balance is converging. For the analysis to stop, it has to reach the iteration limit or meet the convergence tolerances, time, pressure, and section.

The second result to review is the volume change plot as shown in Figure 34. This plot shows in a percentage how much the volume has changed from the original. You would like negative numbers because this indicates a volume reduction. A zero percent change indicates runners that were fixed and not allowed to change.

```
Balance Target Pressure        70.0000 MPa
Mill Tolerance          0.1000 mm
Maximum Iteration Limit   20
Time Convergence Tolerance        5.0000 %
Pressure Convergence Tolerance        5.0000 MPa
Section Convergence Tolerance        0.7000

Iteration    Time Imbalance    Pressure Imbalance  Section Imbalance
                   (%)                (MPa)
  0          21.3837            17.3280            0.6160
  1           1.1076             6.2320            0.3364
  2           2.6103             5.6440            0.3224
  3           1.5539             5.5660            0.3094
  4           0.1441             5.7650            0.2930
  5           1.7397             4.6430            0.2674
Ideal Balance Complete: Allowing for mill tolerance and pressure
control
  6           1.7397             4.6430            0.2674
```

Figure 443: Runner balance screen output

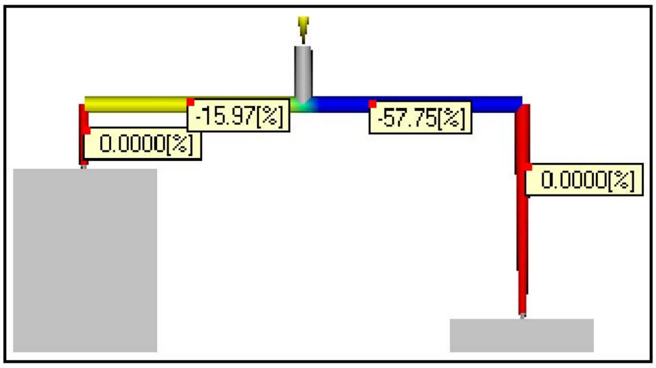


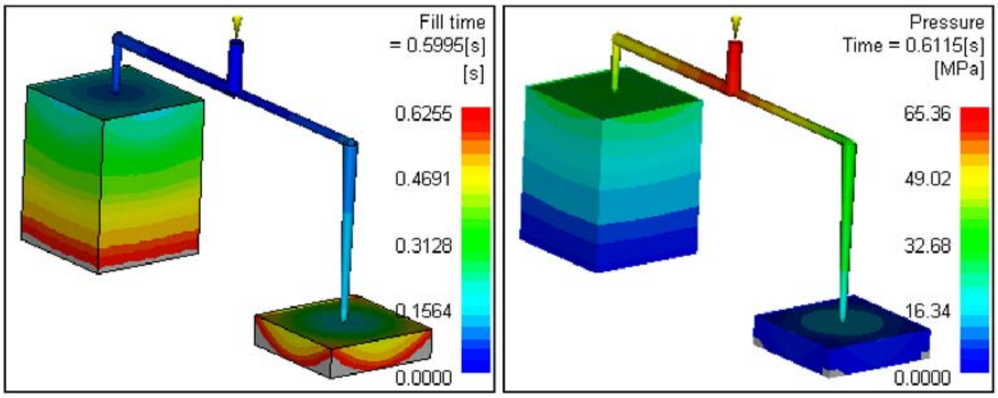
Figure 444: Volume change plot

Second study results

The three main results to review are:

- Time balance.
  - The time imbalance is viewed with the fill time plot as shown in Figure 445A. You know how far out the balance is from the screen output file. The fill time plot allows you to see this graphically. Most of the time, you will want to scale the time results, and possibly use contours to plot the result. The balance here is about 1.7%. The tolerance is 5%.
- Pressure balance.
  - Pressure is far more sensitive to balance than fill time. You should look at the pressure plot as in Figure 36 to determine when the cavities are no longer balanced and compare that time to the end of fill time. The less time difference the better. InFigure 445B the time step used to show the pressure is the last step before the balance is not uniform and is less than 0.02 seconds before the end of fill.

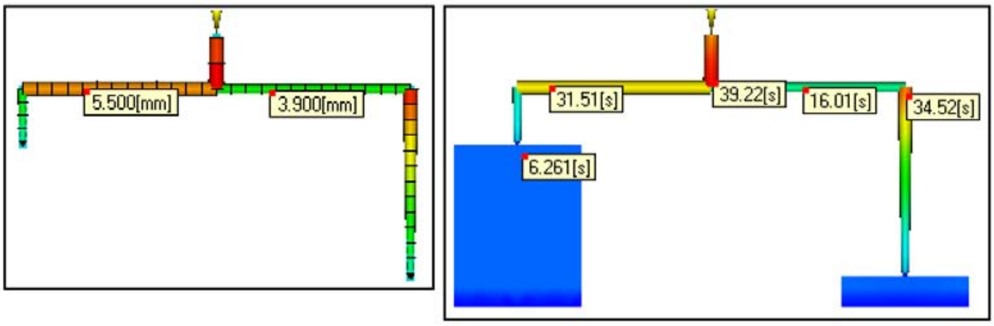
- Runner sizes.
  - If the balance is acceptable with regards to time and pressure, the actual diameters of the runners should be displayed. The diameters could be too large or too small. Figure 446A shows the revised runner diameters of this tool. The original size was 6.0 mm, so there was a reduction in diameter saving material. To determine if the runner is too small, the time to freeze plot should be viewed as shown in Figure 446B. If the minimum runner cooling time is less than 80% of the part cooling time the runner is too small. A conservative approach would put the minimum cooling time at 100%. In the case of this runner system, the runners are not too small. The base of the sprue and the top of the drop on the right of a long cooling time compared to the part. These sections may control the cycle time. The sprue could be made smaller, but the drop would be difficult due to the length and included angle of the drop.



Time imbalance | **Pressure imbalance**

A | **B**

Figure 445: Time and Pressure imbalance



Runner diameters calculated by the runner balance | **Freeze times of the part and runners**

A | **B**

Figure 446: Runner diameters and freeze times calculated by the runner balance

Revise the target pressure as necessary and re-run

Because determining the optimum target pressure may require a number of iterations, the runner balance may need to be run several times. Based on the results from the runner balance analysis just run, set a new target pressure and run the analysis again.

Round the runner sizes if desired

Once you are happy with the runner balance results, you can set the runner sizes to a standard size, or just a more reasonable dimension. Once you manually change the runner diameters, the balance is not optimum anymore. At least one more fill analysis should be run to determine if the balance is still acceptable.

In addition to a filling analysis a packing and warpage analysis can be run to validate the quality of the balance. With a packing analysis, the volumetric shrinkage between the parts should be very uniform. For warpage, linear dimensions and the warp shape should be uniform.

> Runner balancing can be a long process due to the iterative nature of the problem. The large number of elements in multi cavity models makes iterating time consuming.
>
> Runner balancing can be made significantly quicker by reducing the number of elements in the model. This can be done be replacing the cavity with a very simple rectangular beam plate model. This plate model must represent the nominal thickness of the part, the pressure drop and volume of the real part. This equivalent model can be made with 20 to 50 elements rather than thousands of elements in a typical model. The equivalent model must be manually created which might take a few iterations to get it close, but once it is made the runner balance can be done on a very small model. Many analyses with different balance pressures can be run in a very short time. Once the runners are sized, the sizes can be transferred to the full model, a regular flow analysis can be run to verify the runner sizes work.

What You've Learned

Gates come in two different categories including manually trimmed and automatically trimmed gates. Manually trimmed gates include:

- Edge gate.
- Tab gate.
- Sprue gate.
- Diaphragm gate.
- Ring gate.
- Fan gate.
- Flash gate.

Automatically trimmed gates include:

- Submarine gate.
- Cashew gate.
- Pin gate.
- Hot Drop.
- Valve Gate.

There are many rules that govern gate sizing including the material, part geometry, and part application. With simulation gate sizing can also be done by shear rate. If the shear rate is too high, the polymer may degrade. When possible, keep the shear rate as low as possible. A shear rate of 20,000 1/sec. is desirable, but not often practical due to tooling or other concerns.

When designing the runner system, several basic configurations are possible for multi cavity tools, including:

- Herringbone (Standard).
- Geometrically Balanced ("H" Pattern).
- Radial.

When possible, the runner layout should be balanced.

Runner sizes used for a tool depend on the material, flow length, and pressure requirements. When possible, the runners should be minimized in size to keep the runner volume lower.

Runner systems can be created in Synergy using automatic or manual techniques. The automated techniques use the Cavity Duplication and Runner System Wizards. Manual methods involve modeling curves them meshing them or creating beams directly. The Length to Diameter ratio of the beam elements should be 2.5:1, and there must be at least 3 elements defining the gate.

When using either method the part(s) should be moved to tool position. This defines the XY plane as the parting plane, with the positive-Z direction pointing towards the barrel of the machine. Defining the tool position is mandatory for using the wizards, and is useful when using manual methods.

Runner balancing should be done to ensure all the parts fill at the same time and to minimize runner volume. The basic procedure for doing a runner balance includes:

1. Optimize the part.
2. Model the runner system.
3. Run a fill analysis with the runner system.
4. Determine the target pressure.
5. Run the balance analysis.
6. Review the results.
7. Revise the target pressure as necessary and re-run.
8. Round the runner sizes if desired.

Theory and Concepts - Molding Window Analysis

Molding Window Benefits

The molding window analysis is a powerful tool because it can answer some fundamental questions quickly and easily. Most analyses take under one minute to run. The preliminary information derived from a molding window analysis can then be used for further analysis work such as a filling and packing analysis.

Questions That Can Be Answered

The molding window and be used to investigate many of the following questions.

Will the part fill?

This is a fundamental question. Even if you ran the gate location analysis which will look at pressure to some degree, the molding window analysis will look at the pressure to fill a part, in much more detail. The target pressure to fill the part in most cases should be half the machine capacity or less.

> The pressure to fill a part (no feed system) should be about half of the pressure capacity of the machine.

What is the number and basic position of the gates?

If the gate location(s) are not finalized, a molding window analysis can quickly investigate several different gate locations. An analysis is run with each gate configuration and the effect on pressure, shear stress, temperature, etc. can be compared at these different locations.

How big is the molding window?

This is the main reason for running the analysis. The analysis will calculate recommended conditions and will also allow you to see what happens when processing conditions change i.e. mold temperature, melt temperature and injection time. To a large extent, you are answering the first two questions by this third question. Your molding window will be very small if the pressure is too high. One of the best ways to reduce the pressure is to add gates so the flow length gets shorter. The larger the molding window, the less likely you will have problems molding the part.

What material will work best?

The molding window analysis is a quick way to look at how different materials will affect the part. See how the pressure requirements change or how shear stress differs between the materials. The molding window analysis is an excellent tool for evaluating different materials.

Can the part wall thickness be changed?

On midplane and even Fusion models, the thickness of the part can be changed then a quick molding window analysis can be run. You can quickly see how decreasing the wall thickness of a part will require more pressure to fill, a faster injection time and high mold and melt temperatures. You can use the molding window analysis as the first step in deciding to decrease the wall thickness of your part.

What is the part's cooling time?

The part's cooling time is an important part of the cycle for the part. With a molding window analysis, you will be able to quickly see the influence of the part's wall thickness, material, and processing conditions on the cooling time.

Molding window analysis inputs

To run a molding window analysis, the following inputs are required:

- Model, midplane or Fusion.
- Injection location.
- Material.
- Process settings.

Model

The model used for the molding window is the exact same model used for a gate location analysis or a regular flow analysis; therefore, the mesh should not have any mesh errors in it. Due to the assumptions of the molding window analysis runners and gates must NOT be part of the model. Only the plastic part itself should be modeled.

> The model must NOT contain any part of the feed system when running a molding window analysis.

Injection location

One or more injection locations must be defined for the part. Once the injection location(s) has been selected, the molding window analysis can simplify the part by using a 2D model to create the molding window. The 2D model represents the thickness, length and volume of the part, making it a suitable model for investigating the process settings for the part.

Material

A thermoplastic material must be selected just like any other analysis.

Process settings

To run the molding window analysis, you have the option to select the specific molding machine, mold and melt temperature ranges and injection time to be analyzed. The default values can be used, however the analysis results may be easier to interpret if some process settings are changed.

Possibly, the injection pressure capacity of the molding machine should be changed. The default value is 180 MPa (~26,000 psi). This value is higher than a typical molding machine, and could be lower than some. If you don't know the pressure capacity of a molding machine, a safe assumption is 140 MPa (~20,000 psi). Also, the pressure limit factors can also be set so the entire molding machine pressure is not used.

The default mold and melt temperatures are the minimum to maximum ranges specified in the material database. In rare occasions the ranges analyzes is adjusted. The preferred molding window determines the injection time range analyzed, and is explained later in the module.

Running a Molding Window Analysis

The process of running a molding window analysis are interpreting the results is shown in Figure 327. These steps are explained in detail in the following sections.

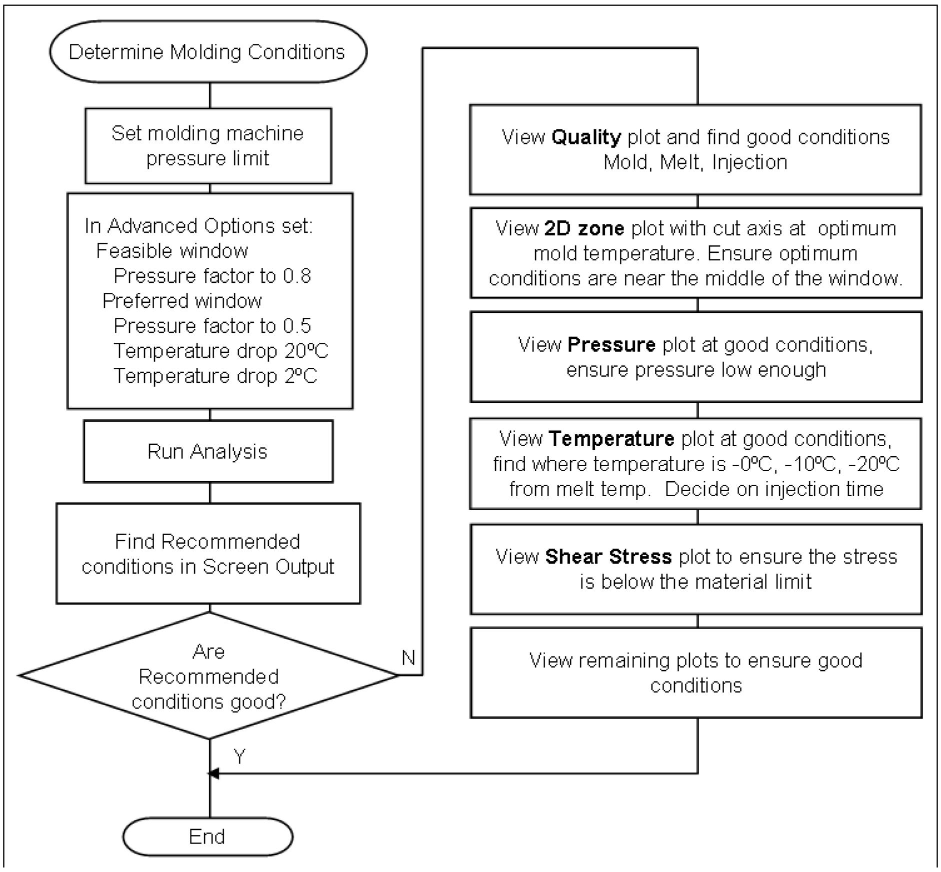


Figure 327: Flow chart of the molding window analysis

Process settings

A molding window analysis can be run with default settings in the Process Settings Wizard, but to make the results more meaningful and easier to interpret, defaults should change.

Injection Molding Machine

A molding window analysis can be run without changing any of the default process settings. However, as mentioned above, the default molding machine pressure is 180 MPa. Not changing the maximum pressure to the limit of the molding machine only influences one result, the zone plot, which graphically shows the size of the molding window. If the pressure is not changed all you need to do is take that into consideration when looking at results.

To change the value of the machine's injection pressure limit:

4. Click the **Edit** button, on the Molding window settings page in the Process Settings Wizard.
5. Click the **Hydraulic Unit** button on the Injection molding machine dialog.
6. Enter a new Machine pressure limit.
7. Click **OK**.

Process Parameters

The defaults for the mold temperature, melt temperature, and injection time are all set to **Automatic,** as shown in Figure 328. For the temperatures, this is the range as specified in the material database. The automatic injection time is determined during the analysis. A suitable range of injection times are calculated. For all for the process parameters, a range can be specified.

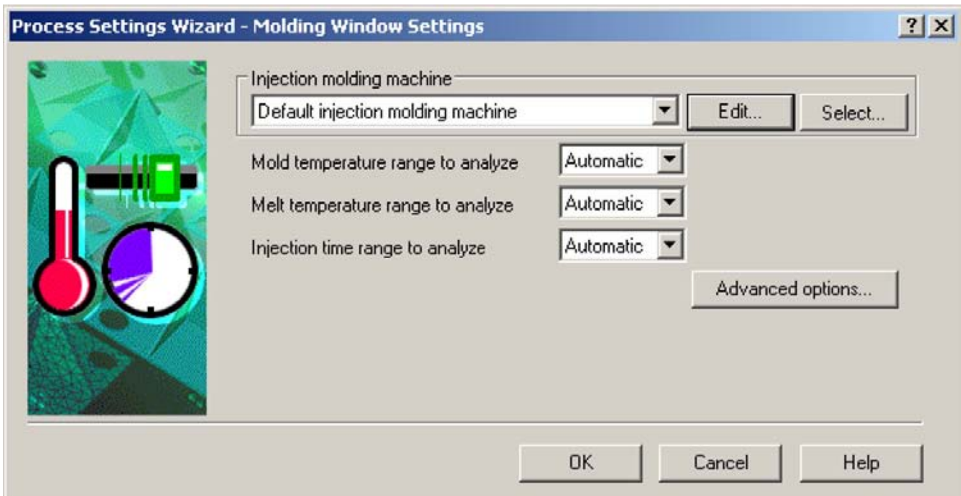


Figure 328: Molding window process settings

Advanced Options

The **Advanced Options** accessed by the Advanced Options button on the **Process Settings** dialog. The advanced options influence the calculations of the **zone** plot. The zone plot uses green, yellow and red to indicate the how moldable a part is at a given set of conditions. The Zone plot will be discussed later.

In the Advanced Options, as shown in Figure 329 there are two groups of settings. One is a **feasible molding window** and the other is a **preferred molding window**. The feasible only has one variable set, by default; the injection pressure limit with a factor of 1. This means a feasible window will be anything under the limit of the machine.

This is where the pressure limit of the molding machine becomes important. If the actual molding machine has a pressure limit of 140 MPa (20,000 psi) and the pressure limit set for the molding window analysis is the default 180 MPa (26,000 psi), then the zone plot result will indicate that the pressure limit has not been reached but in fact it may have been exceeded. It is not critical that the injection pressure limit of the molding press be known and set, but if it is, the Zone plot will be easier to interpret.

An option would be to set the Injection limit pressure factor to less than one. This would warn you when the pressure to fill you part is approaching the limit of the molding machine. A factor of 0.8 is a reasonable value.

Moldflow recommends that the pressure limit of the molding machine be set to 140 MPa (20,000 psi) if the actual molding machine pressure is not known.

The Injection Pressure Limit Factor may be set to 0.8. This will warn you when the filling of your part is approaching the limit of your machine.

The preferred molding window is a molding window that makes a good part. In Table 43 below, the variables, the default settings, and result interpretation criteria are listed.

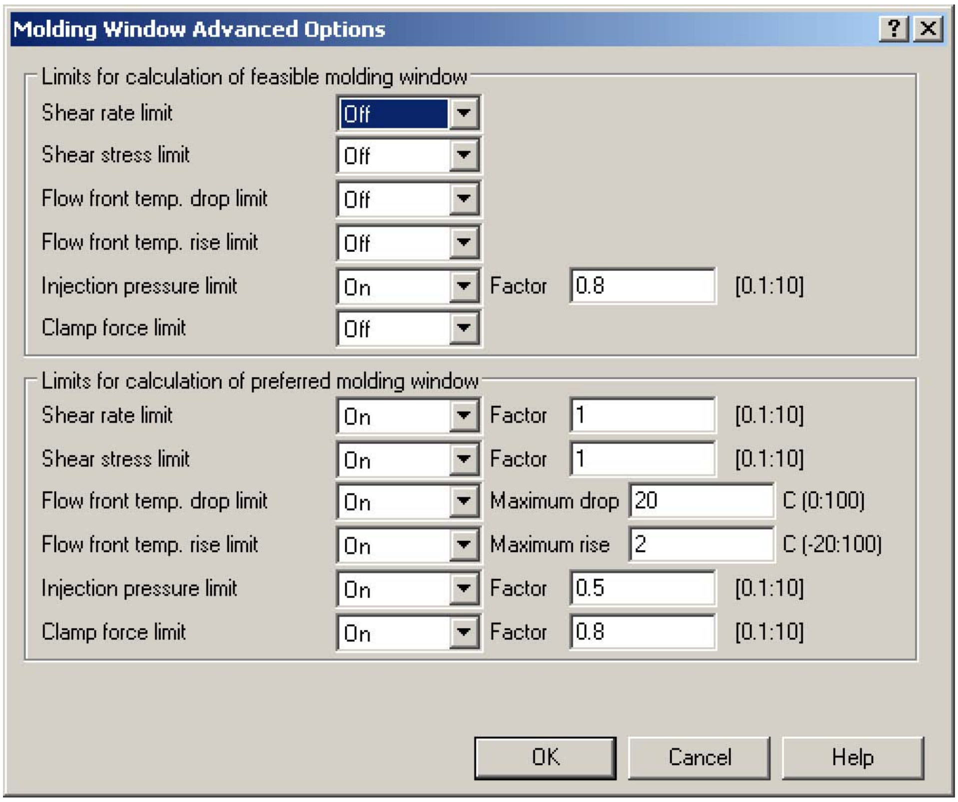


Figure 329: Molding window advanced options

Table 43: Preferred molding window criteria

Variable	Default	Interpretation
Shear rate	1	The maximum shear rate on the part should not be above the limit for the material as stated in the material database. This should never a problem.
Shear stress	1	The maximum shear stress on the part should not be above the limit for the material as stated in the material database. This can sometimes be a very big problem depending on the type of material or application. It is possible to have no preferred molding window because the shear stress is always too high. If the part will be used in a harsh environment the shear stress limit factor should be lower somewhere between 0.5 and 0.8. The shear stress shown is the maximum found. If there is no preferred window due to shear stress, a fill analysis should be run with the best conditions excluding the consideration of shear stress. You may find that most of the part has a shear stress limit that is acceptable, and only a small non-critical area has high stress.
Flow front temperature drop limit	10° C, or 18° F	The flow front temperature drop is the most limiting variable. Up to 20°C. temperature drop is generally acceptable for most materials and applications. The default value of 10 °C is conservative. You can keep the default value, or change it up to 20°C.

Table 43: Preferred molding window criteria

Variable	Default	Interpretation
Flow front temperature rise limit	10º C, or 18º F	The flow front temperature is allowed to rise, but the highest quality is a zero degree rise. Generally only a rise of 2ºC is acceptable. It is best to lower the allowable temperature rise.
Injection pressure limit	0.8	How this value is set will depend on what you have set for the injection molding machine pressure limit and the feasible molding window pressure limit. Moldflow recommendations include setting the injection molding machine pressure limit, and the feasible molding window pressure factor to 0.8. When this is done, the preferred molding window pressure factor should be set to 0.5. The pressure to fill the part should be no higher than 50% of the machines pressure capacity. The molding window analysis is only looking at the part, not the runner system. You need to leave room for pressure drop through the runners and still not reach the capacity of your machine. Setting this value to 0.5 as the factor is a conservative value and is recommended.
Clamp force limit	0.8	The limit is 80% of the machine as generally the highest clamp force is during packing. The default molding machine has a clamp tonnage limit of 7000 metric tons so this limit is unlikely to be reached unless a specific machine is used. Clamp force issues are best addressed with a full flow analysis.

The molding window analysis is designed for use without a runner system. The guidelines for determining a preferred window do not work well with runners due to the shear heat buildup.

Molding Window Analysis Interpretation

This section includes a general discussion of molding window result interpretation. Interpretation of an actual analysis will be done in the practice section.

Results from a molding window analysis are in three forms:

- Log files.
- Zone plots.
- XY graphs.

General Interpretation Procedure

The procedure for looking at molding window results will vary, depending on the objectives of the analysis. The basic procedure is as follows:

1. **View the screen output log.**

   1.1. Find the recommended processing conditions near the bottom of the screen output log, as shown in Figure 330 on page 485.

   1.2. Compare the recommended conditions of mold and melt temperature to the ranges near the top of the screen output.

   - ➤ The recommended conditions should preferably be near the middle of the ranges.
   - ➤ If not, this would indicate you might need to investigate choosing conditions other than the recommended conditions.

2. **View the Quality XY Plot.**

   - The molding window analysis determines the recommended processing conditions based on the set of conditions that has the highest quality. The quality is calculated using the parameters in the advanced options described earlier. The maximum quality possible is 1.0. Generally the maximum values for a given part are between 0.8 and 0.95.

   2.1. View the quality plot and set the X-axis of the graph to be injection time using the plot properties, as indicated in Figure 332 on page 486.

   2.2. Adjust the mold and melt temperature slider to the recommended mold and melt temperature found in the screen output.

   2.3. Normally, the data point with the highest quality is easy to visualize, as there is a sharp drop in quality from the maximum.

   2.4. Move the mold and melt temperature scroll bars and see how the quality is affected.

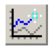
   - ➤ In most cases, there will be many sets of conditions that the quality is similar to the maximum.

   2.5. If the recommended conditions are not practical because, for instance, the recommended melt temperature is at the minimum value, move the slides to a mold and melt temperature you want to investigate.

   - ➤ Generally this is in the middle of the ranges, and with a high quality.

   2.6. Query the maximum quality at the mold and melt temperature you want to investigate further. This will be the injection time, as indicated in Figure 333 on page 486.

   - ➤ The mold temperature, melt temperature and injection time are now the new conditions and need to be investigated further.

3. **View the Zone (molding window), 2D Slice Plot.**
   - The 2D zone plot shows a 2D shaded graph. It uses mold temperature, melt temperature and injection time. Two of the 3 variables form the axis of the graph, the other is the **Cut axis** for the graph. The best cut axis is mold temperature. When the cut axis is mold temperature, the X-axis is melt temperature, and the Y-axis is injection time, as shown in Figure 331 on page 485.
   - The cut axis can be animated with the  icon.
   - The 2D zone plot will give you a sense for the size of the molding window. The plot uses 3 colors:
     - **Green** - represents an areas that is within the preferred molding window.
     - **Yellow** - represents the size of the feasible molding window. This would mean that one of the parameters of the preferred window is outside the limit. More than likely it is a temperature limit.
     - **Red** to indicate the pressure is higher than the factor set for the feasible molding window.

   3.1. Set the cut axis to Mold temperature and set the value to the mold temperature found in the quality plot.

   3.2. Query the zone plot to find the injection time and melt temperature found in the quality plot.
     - Ideally, the chosen conditions are near the middle of a large preferred window.

> The zone plot is most meaningful if the machine pressure capacity is known and the pressure factors on for the feasible and preferred windows are set.

4. **Check the Pressure XY Plot.**

   4.1. Set the X-axis to be injection time, using the plot properties, and slide the mold and melt temperature to the new conditions.

   4.2. Use the query tool to find the new injection time on the pressure curve.
     - This represents the pressure required for the new processing conditions. See Figure 334 on page 486.

   4.3. Make sure the pressure is under the 70 MPa (10,000 psi) guideline, or whatever the about half the machine injection capacity is.

5. **Check the Minimum Flow Front Temperature XY Plot.**

   5.1. Set the X-axis to be injection time, using the plot properties, and slide the mold and melt temperature to the new conditions.

   5.2. Use the query tool to find the new injection time.
     - This will be where the minimum flow front temperature is equal to the melt temperature and is where the quality is the highest. See Figure 335 on page 487.

5.3. Use the query tool to find where the temperature is 0º C, 10º C. (18º F), and 20º C. (36º F) below the melt temperature.

- A 0º C drop in temperature defines the highest quality.
- A 10º C (18º F) drop in temperature in most cases is a very acceptable amount of drop.
- A 20º C (36º F) defines the limit of the preferred molding window, assuming it was set a maximum temperature drop of 20ºC (36ºF) in the advanced options.
- Finding at what times these temperatures occur will give you another way to get a sense for the size of the molding window as the flow front temperature is generally the limiting factor in the molding window.

6. **Check the Maximum Shear Stress XY Plot.**

6.1. Set the X-axis to be injection time, using the plot properties, and slide the mold and melt temperature to the new conditions.

6.2. Make sure the shear stress is below the material limit.

- The lower the shear stress, the better.
- Typically, the maximum shear stress plotted in this result will be significantly higher than the nominal shear stress in the part. If the shear stress is near or above the limit for this plot, you should concentrate on the shear stress result when you run a filling or flow analysis. You should find that the majority of the part has acceptable levels of stress and there may be some limited areas of high stress.

> Up to four XY plots can be viewed and manipulated at the same time. Split the screen into 2 or 4 windows. Plot in each of the windows a different XY graph. For one of the graphs, open the plot properties. On the Explore Solution Space –XY Plot dialog, check the Lock all molding window XY plots in this study box. Now as you manipulate the sliders, all windows will move.

7. **Check remaining plots.**

7.1. Cooling time

- The cooling time is viewed to see what affect processing conditions have on the cooling time. Mold temperature has the greatest influence so that should be the X axis.

7.2. Shear Rate

- The shear rate will never be excessive in your part as a whole. There may be some very local areas where the shear rate approaches the limit. Plotting the shear rate from a molding window analysis will show you how the shear rate drops with increases in injection time.

```
Analysis commenced at        Tue Jan 17 17:46:00 2006
Flow is using stored mesh match and thickness data
Match data was computed using the maximal-sphere algorithm
   Mold temperature range to analyze                   =   Automatic
      from mold temperature                            =        80.0 C
      to   mold temperature                            =        95.0 C
   Melt temperature range to analyze                   =   Automatic
      from melt temperature                            =       270.0 C
      to   melt temperature                            =       295.0 C
   Injection time range to analyze                     =   Automatic
   Limits for calculation of feasible molding window
      Shear rate limit                                 =         Off
      Shear stress limit                               =         Off
      Flow front temperature drop limit                =         Off
      Flow front temperature rise limit                =         Off
      Injection pressure limit factor                   =        0.80
      Clamp force limit                                =         Off
   Limits for calculation of preferred molding window
      Shear rate limit factor                          =        1.00
      Shear stress limit factor                        =        1.00
      Flow front temperature drop limit                =       20.00 C
      Flow front temperature rise limit                =        2.00 C
      Injection pressure limit factor                  =        0.50
      Clamp force limit factor                         =        0.80
Maximum Design Clamp Force     7000.22 tonne
Maximum Design Injection Pressure    :      140.00 MPa
Recommended Mold Temperature    :      90.00 C
Recommended Melt Temperature    :     290.45 C
Recommended Injection Time      :     0.4384 s

Execution time
   Analysis commenced at        Tue Jan 17 17:46:00 2006
   Analysis completed at        Tue Jan 17 17:46:04 2006
   CPU time used                         3.08 s
```

Figure 330: Moldingw window screen output log

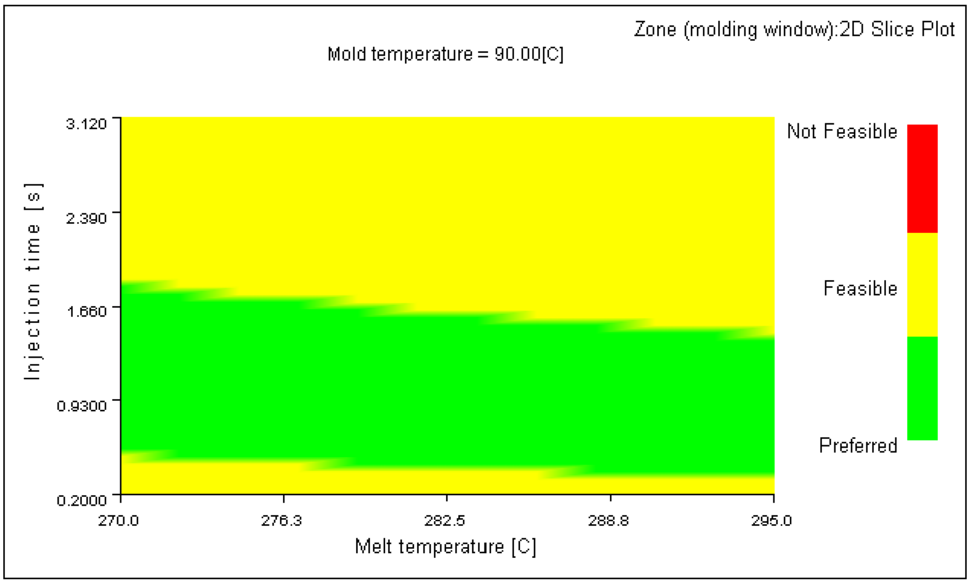


Figure 331: 2D Zone plot showing the size of the molding window

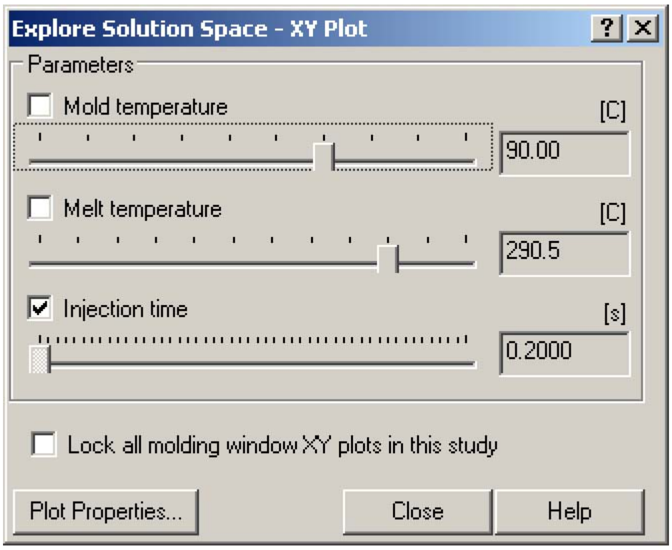


Figure 332: Molding window, explore solution space - plot properties

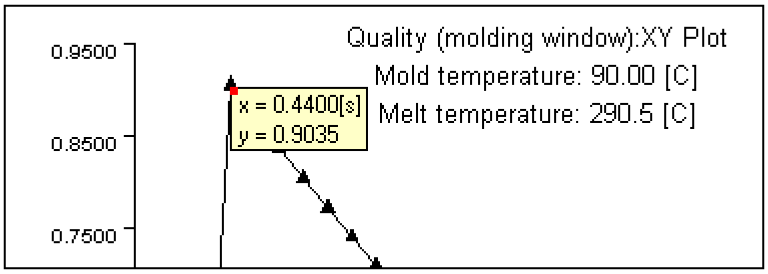


Figure 333: Query highest quality

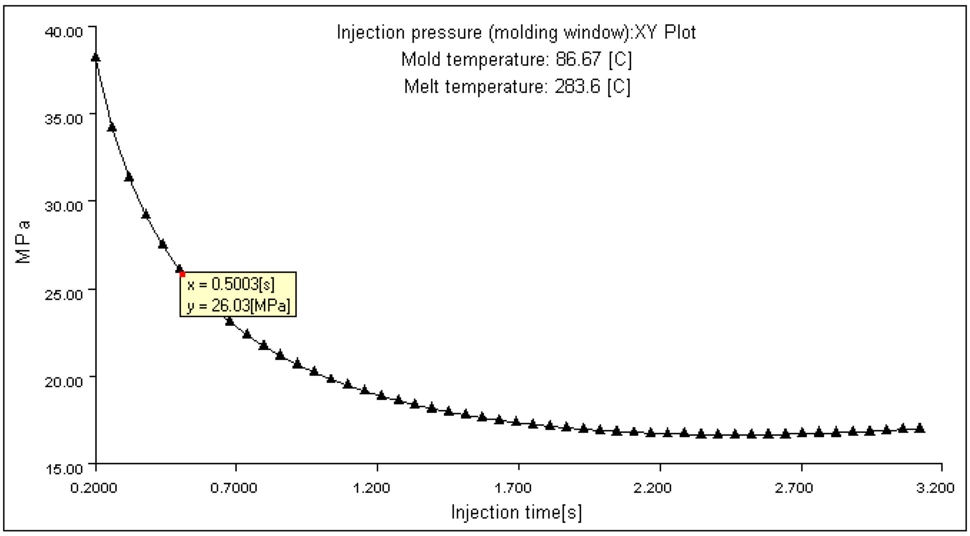


Figure 334: Pressure XY graph

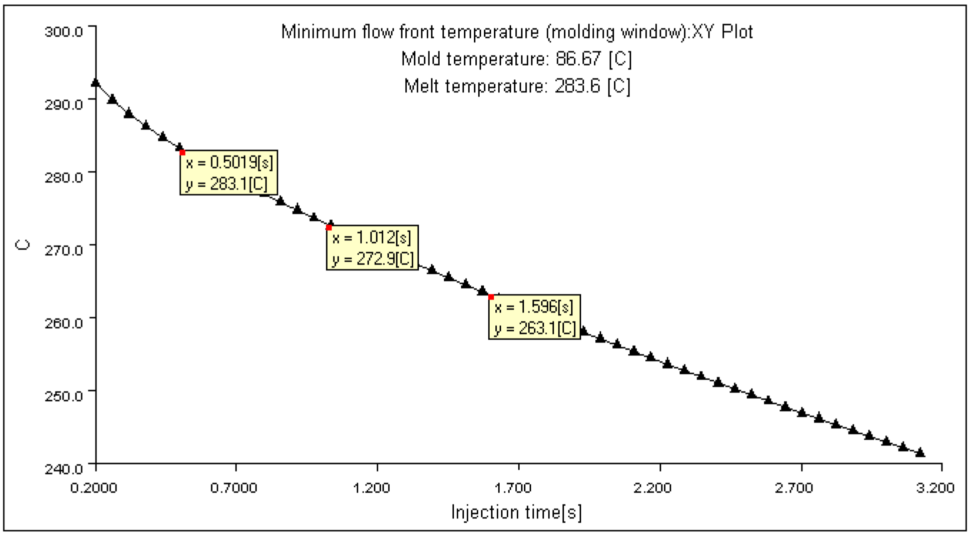


Figure 335: Minimum flow front temperature graph

Answering Questions with the Molding Window Analysis

The discussion above concentrated on the interpretation of the molding window results and answered several of the fundamental questions that were discussing in the beginning of this chapter, including:

- Will the part fill?
- How big is the molding window?

For the question "**What is the number and basic position of the gates?**" we know that the gating used the fill pressure is within acceptable limits because there was a rather large molding window. In this case, additional gates are not needed. If the pressure drop were quite high the number and/or position of the gates should be changed to shorten the flow length.

Additional questions and be answered also, including:

- What material will work best?
- Can the part wall thickness be changed?
- What is the part's cooling time?

Examples of using a molding window analysis to answer these questions are below. The part used is a door panel shown in Figure 336.

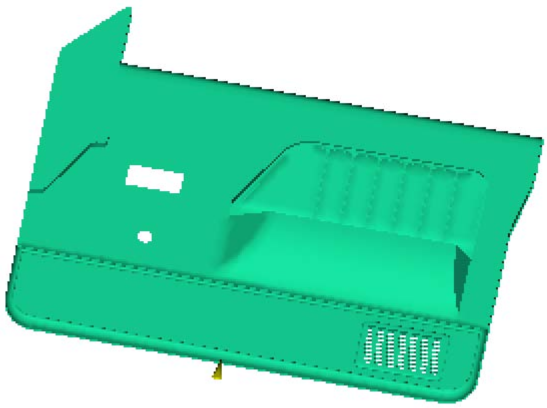


Figure 336: Door panel used for molding window examples

Investigating the number of gates

The molding window analysis can help you determine the number of gates on the part. If one gate requires a pressure that is too high, a second gate should be added. When adding a second gate, keep in mind the design rules for adding gates, primarily, the flow length from each gate must be the similar and the filling must be balanced. In this example, one gate was in the center of the bottom edge of the door panel. The multiple gate example has 5 gates along the bottom edge. Figure 337 shows the Pressure vs. Time curve for a single and multi-gated part plotted together. This was done in a spreadsheet. You can see that the multiple gates have a lower pressure at all injection times compared to a single gate. In this case, the reduction is not significant, because the flow length was not reduced too much.

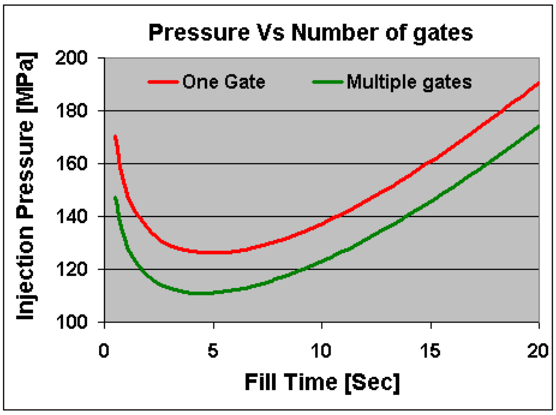


Figure 337: Comparing injection pressure vs. the number of gates

Investigating different materials

The molding window analysis can easily be used, to see the effect of two different materials. Simply run a molding window analysis for each material and compare the results. Normally, the most useful result will be the injection pressure. Normally the material you compare will be of the same type so set the mold and melt temperature to the same values and look to see how the pressure changes over time. Figure 338 shows the pressure vs. time graph for two grades of ABS, from the same supplier. Notice how their pressure curves are a bit different.

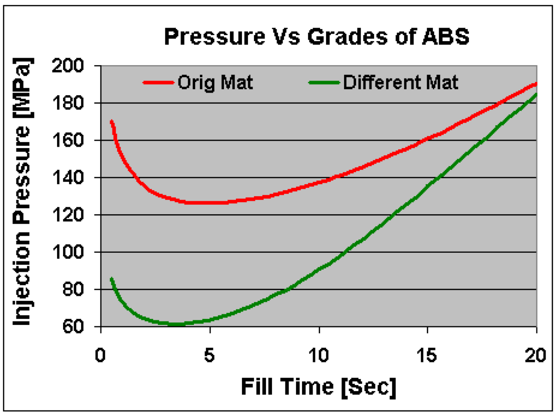


Figure 338: Comparing different materials with a molding window analysis

Investigating Different Wall Thicknesses

The molding window analysis is an excellent tool for some initial investigation on the wall thickness of a part. Figure 339 shows a case where the part wall thickness was increased to see if the pressure could be lowered enough. Normally, the problem would be the other way, you would like to know how much thinner you could make the part and still fill it.

In this preliminary investigation, a Fusion model was used. The thickness property of the elements were changed. If the decision was made to change the thickness, the solid model for the part would need to be revised to change the wall thickness and the part re-imported. You can't run a cooling or warpage analysis with the thickness property globally changed to a different thickness. The cooling analysis works on the actual location of the elements not the thickness property.

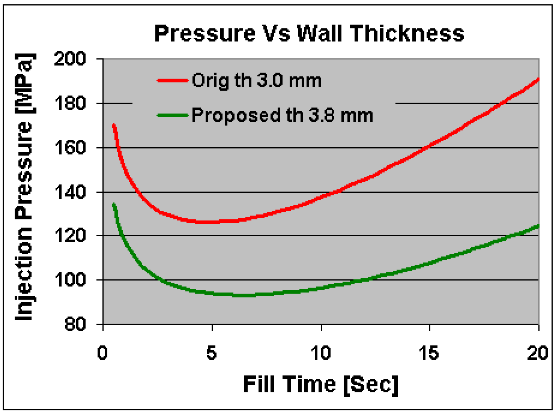


Figure 339: Comparing different wall thicknesses

Investigating Cooling Time

How fast the part cools is very dependant on not only the part geometry, but the mold temperature and type of material. Below in Figure 340, the cooling time vs. Mold temperature is plotted for two materials. The thermal properties can make a big difference in how the part cools. You can see at higher mold temperatures, the cooling time of the different material crosses the original material.

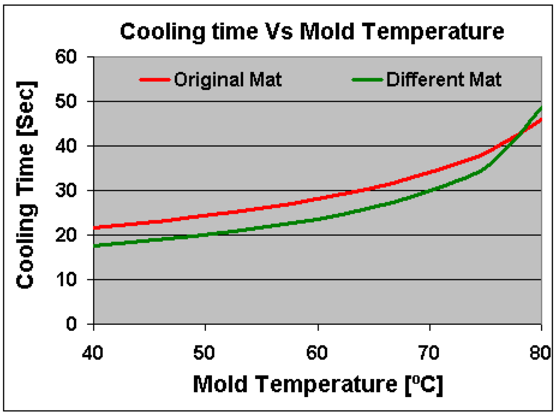


Figure 340: Cooling times for two materials

Summary

Figure 341 shows the 2D zone plot for the four analyses that were used to produce the graphs in Figure 337 to Figure 339. Each analysis was run with the same mold temperature range (230ºC to 260ºC) and the same injection time range, (0.5 sec. to 20.0 sec.).

The upper left zone plot is the original design and is the one all others are compared to. This plot has no preferred area because the injection pressure is too high.

The upper right zone plot represents the part using multiple gates. Here there is a small preferred area. The flow length was not shortened much so the pressure did not fall very far.

The lower left plot has the largest preferred area. This was done by increasing the wall thickness from 3.0 mm to 3.8 mm. Now the part can be filled with less than half the machine capacity for the entire temperature range.

The lower right plot uses the original gate location, but with a material of much lower viscosity of the same type, (ABS). The preferred area is a has a narrow time range and is at a much lower time than with the original material due to the thermal properties of the material.

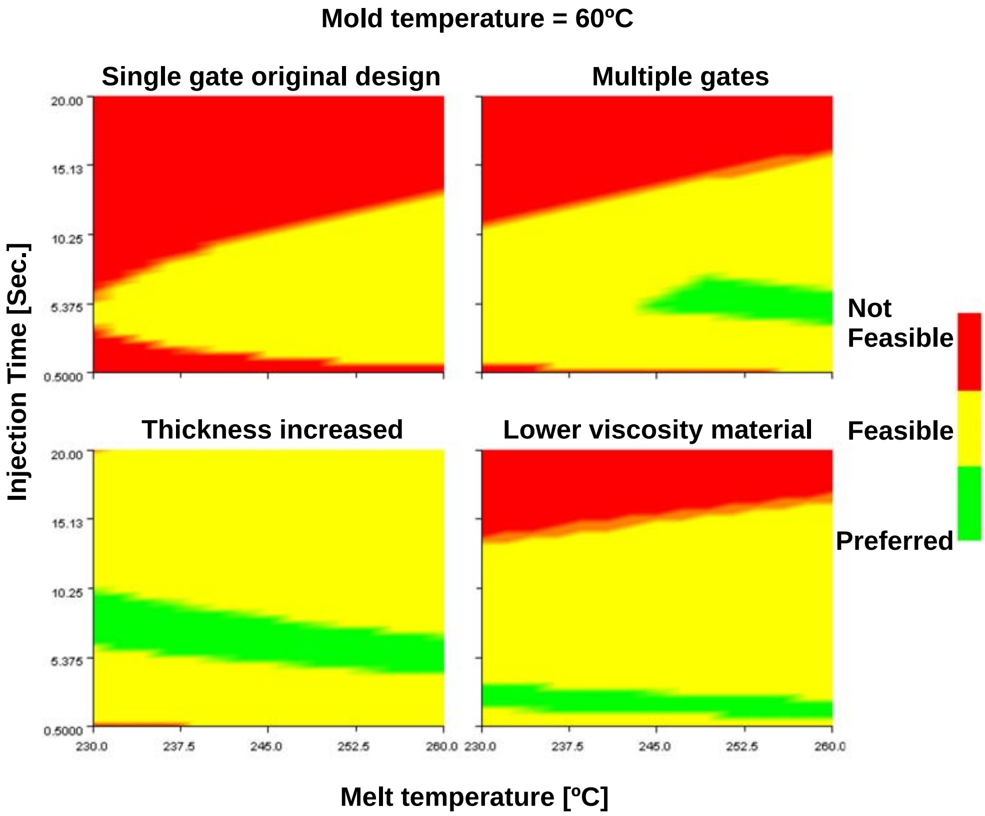


Figure 341: 2D Zone plot comparing the 4 examples from Figure 337 to Figure 339

What you've learned

The Molding Window analysis is a quick and easy way on a midplane or Fusion model to determine initial molding conditions. Questions that can be answered in addition to the molding conditions include:

- Will the part fill?
- What is the number and basic position of the gates?
- How big is the molding window?
- What material will work best?
- Can the part wall thickness be changed?
- What is the part's cooling time?

Molding window analysis inputs include:

- Model, a midplane or Fusion model, of the part only, (no runners)
- Injection location, one or more injection locations defined directly on the part.
- Material, the thermoplastic material that will be evaluated.
- Process settings, including:
  - Injection Molding Machine, the injection pressure capacity of the molding machine should be set.
  - Process Parameters, The range of mold temperature, melt temperature and injection time to be analyzed must be defined. The default is automatic which is fine most of the time
  - Advanced Options, allows the user to adjust how the molding window needs to be defined, normally the acceptable pressure and temperature ranges must be set.

Molding window analysis interpretation involves looking at several results to determine the recommended conditions, then possibly modifying them. The steps include:

1. Look at the screen output log file to find the recommended conditons.
2. View the quality plot to decide if the recommended will be kept or not.
3. View the 2D zone plot to see the size of the molding window and to find the location of the processing conditions.
4. Plot the injection pressure to ensure the required pressure is below the machine capacity.
5. Plot the minimum flow front temperature to ensure the injection time produces an acceptable amount of temperature drop.
6. Check the Shear stress plot to ensure the polymer is not over stressed.
7. Check the cooling time and shear rate plots to ensure they are OK.

CHAPTER 18

Basic Packing

Aim

The aim of this chapter is to learn the procedures for running a packing analysis.

Why do it

How well a part is packed out is of primary importance when considering warpage, shrinkage, and defects such as sink marks. The main output of a packing analysis is volumetric shrinkage, and the distribution and magnitude of volumetric shrinkage, which plays a key role in part quality.

Overview

In this chapter you will review:

- When to run a packing analysis.
- Definitions of terms used for packing.
- Input parameters used for a packing analysis.
- Running a packing analysis.
- Reviewing results for all 3 mesh types.

Theory and Concepts - Basic Packing

When to run a packing analysis

A packing analysis is done as portion of a part optimization process, specifically optimization of the flow through the part, as shown in Figure 459. Packing is best done after:

- The part has been optimized for filling.
- The runners have been sized and balanced.
- A cooling analysis has been run.

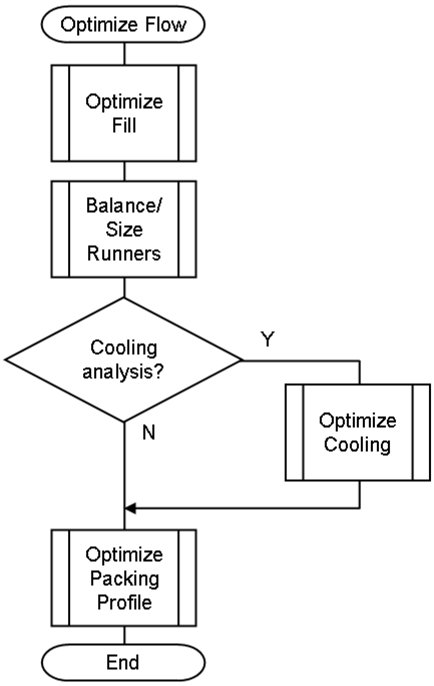


Figure 459: Optimize flow or a part flow chart

Optimized filling

Filling optimization includes all the aspects of the filling of the part that should be addressed early in the product design stage. This includes, the location of the gate(s) and the molding conditions.

Sized and balanced runners

In a typical design process, the runners are sized or balanced after the filling is optimized. The runner system is critical to have modeled for an accurate packing analysis. The size of the gates and even the runners can control how well the part packs out.

Cooling analysis

Running a cooling analysis before a packing analysis is not required, but is highly recommended. During packing, heat transfer dominates the process. The cooling analysis models the efficiency of the tool's ability to extract heat from the part. The heat extraction is never uniform, there will always be hot and cold spots in the tool. Without cooling the heat extraction is assumed to be uniform, therefore the packing analysis will be less accurate when cooling is not used as input. Preliminary work can be done, but the final packing analysis should have cooling as an input. The analysis sequence will be Cool+Flow.

Definitions

There are several terms used in the injection molding industry related to packing. These terms will be discussed in Table 72 in the context of how they are used in a packing analysis. Figure 460 below summarizes the definitions.

Table 72: Packing definitions

Term	Description
Packing Pressure	The magnitude of pressure that is applied after the velocity to pressure control switch-over. (V/P switch-over)
Packing Time	The time pressure is applied after the V/P switch-over.
Hold Pressure	Often used interchangeably with packing time. This can also mean a different magnitude of pressure being applied after the V/P switch-over. Normally this is a lower pressure. On an injection molding machine, there may be one control for packing, then another control for holding. They both apply a pressure for a specified amount of time. In MPI, there is only one control for the pressure applied after the V/P switch-over. It is called **Pack/holding control**. Generally it is called just packing. The pressure can be changed as many times as needed to create the profile necessary. This profile can be transferred to the machine into one or two machine controls.
Hold Time	The time that hold-pressure is applied.
Cooling time	The cooling time is the time after packing when there is no pressure being applied to the part. The part continues to cool until it can be ejected from the mold. Injection time, packing time, cooling time and clamp open time combined define the entire cycle for a flow analysis.

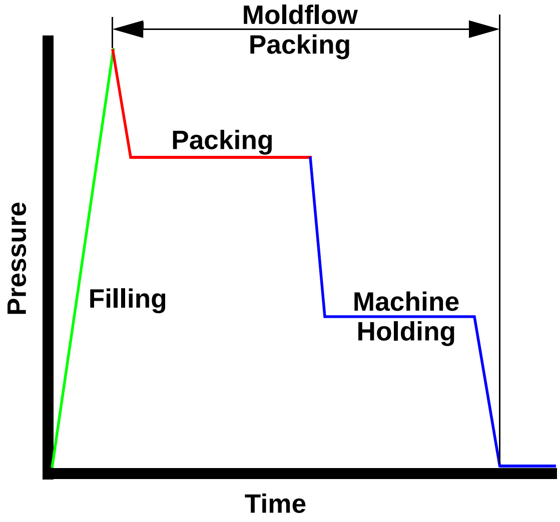


Figure 460: Summary of packing definitions

Packing analysis inputs

There are three inputs required to run a packing analysis in addition to the inputs for a filling analysis. The inputs are:

- Packing time.
- Packing pressure.
- Cooling time.

A block profile is entered in a table, as indicated in Figure 461 below. The left column is a time in duration and the second column is pressure.

Packing pressure vs time

	Duration s [0:300]	Packing pressure MPa [0:500]
1	0	36
2	8.8	36
3	0	0

Figure 461: Block profile in the packing pressure vs. time input table

The profile in Figure 461 is read: "**It takes 0 seconds to get to 36 MPa** (From the pressure at V/P switch-over)**, and then takes 8.8 seconds to get to 36 MPa**." Since the pressure is at 36 MPa, the pressure is maintained for 8.8 seconds.

The dialog in Figure 462 is set up to have a decayed profile:

Packing pressure vs time

	Duration s [0:300]	Packing pressure MPa [0:500]
1	1	36
2	8.8	36
3	1	0

Figure 462: Decayed pressure profile

In this case, the pressure will take 1 second to get to 36 MPa from the V/P switchover pressure and another 1 second to get to 0 pressure. Figure 463 graphically shows both profiles.

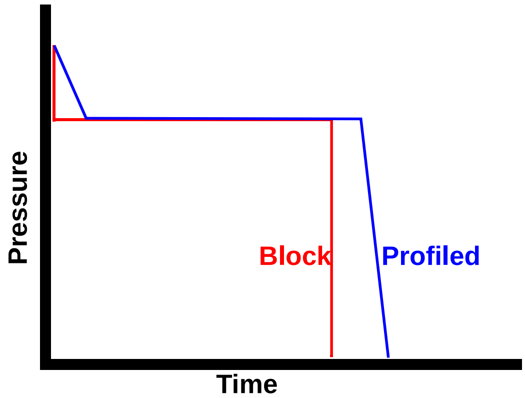


Figure 463: Example profiles

Packing profile methods

The packing profile can be entered 4 different ways including:

- % Filling pressure vs. time, (default).
  - Instead of putting in a pressure directly, the packing pressure entered is a percentage of the fill pressure. The default profile is 80% of the fill pressure for 10 seconds.
- Packing pressure vs. time.
  - This is the method described in the three previous figures.
- Hydraulic pressure vs. time.
  - The difference between packing pressure and hydraulic pressure is the machine's intensification ratio. For most machines, it is about 10:1. The packing pressure is 10 times that of hydraulic pressure. Most of the time, packing pressure is used to input a profile if the default% Filling pressure is not used.
- % Maximum machine pressure vs. time.
  - Rarely is this method used. If the machine controller specifies the packing as a percentage of the machine's maximum pressure and an existing process is being duplicated, this method is used.

The cooling time on the main page of the Process Settings wizard and can be entered as a specified time, or automatic. Figure 464 shows the cooling time settings both as specified and automatic. The automatic method will calculate the cooling time so that 100% of the part is frozen based on the ejection temperature on the database. The user can specify the ejection temperature or the percentage frozen by clicking on the Edit ejection criteria button.

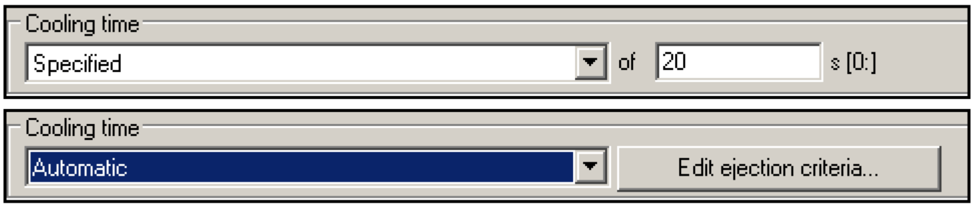


Figure 464: Methods of cooling time input

Determine packing pressure

The packing pressure used for packing out a part is often related to the fill pressure. As a rough guideline to get started, the packing pressure should be about **80%** of the fill pressure. This is the default value; however, the packing pressure can vary significantly. Packing pressures are common between 20% and 100% of the fill pressure, and can be higher and lower than this range.

An important aspect of the packing pressure is that it can't be high enough to exceed the clamp limit of the machine. The formula below is used to estimate the maximum pressure that should be used. This formula will determine a pressure assuming a constant gradient across the part so that 80% of the machine capacity will be used. This is a conservative approach since the pressure is never uniform across the part and the limit is 80% of the machine capacity. However, this can be used as a starting point.

$$P_{max} = \frac{F}{A}100 \times 0.8$$

$$\begin{aligned} P_{max} &= \text{Maximum packing pressure that should be used} \\ F &= \text{Machine clamp force limit (tonnes)} \\ A &= \text{Total projected area of the model (cm\^{}2)} \\ 100 &= \text{Unit conversion} \\ 0.8 &= \text{Safety factor, use 80\% of maching capacity} \end{aligned}$$

Determine packing time

The packing time should be long enough so the gate freezes off. This time can be estimated from the time to freeze plot from a filling analysis; however, it will generally be low due to shear heat during packing. It is best to estimate a very high value for the first packing analysis, for example, a 20 second packing time can be set if you think the gate freezes off in 4 seconds. In the next packing analysis, the packing time can be reduced to a more reasonable value.

> The best way to determine the gate freeze time is with the frozen layer fraction plot and see when the gate is frozen. This can only be done after an initial packing analysis so the an initial estimate is needed.

Running a packing analysis

The packing analysis is run when the analysis sequence is set to **Flow**. A fill analysis only calculates until the part volume is 100% filled. With the analysis sequence set to Flow, the filling is done, then the packing is completed.

The only additional input required for a Flow analysis compared to a fill analysis is the cooling time. The switchover and packing profile are needed for the filling analysis because the part switches from the velocity controlled filling and pressure controlled packing before 100% of the cavity volume is filled.

Velocity/pressure switchover

You may find that the default values for the **velocity/pressure switchover** may need to be changed. The default value is "**automatic**" and calculates the switchover such that if the ram stopped instantaneously, there would be enough decompression of the polymer to just fill the cavity. Once the switchover is calculated with the automatic method, another method is used for additional analysis work. Other methods of switchover that can be used include:

- % volume filled.
- Injection pressure.
- Hydraulic pressure.
- Clamp force.
- Injection time.

> The most common method of switchover is by%volume filled if automatic is not used.

The percentage the part is filled when using the automatic switchover will depend on the compressibility of the polymer, (it's PVT characteristics) and the pressure required to fill the part.

Pack/holding control

There are four methods of Pack/holding control used, including:

- %Filling pressure vs. time.
  - This is the default setting, with a percentage of 80% for 10 seconds.
  - Use when the filling pressure is not known and clamp tonnage is not an issue.
  - Once initial packing analyses are done, packing pressure is often used rather than a percentage.
- Packing pressure vs. time.
  - This is the pressure applied directly applied to the polymer.
  - Most commonly used when the fill pressure is known and a specific packing pressure is desired.
- Hydraulic pressure vs. time.
  - This setting is rarely used. The Hydraulic pressure times the intensification ratio equals the packing pressure.
  - This setting is used if the specific molding machine is set, and an existing process is being duplicated.
- %Maximum machine pressure vs. time.
  - This setting is rarely used. The specific molding machine must be specified and its pressure capacity defined.
  - This setting is used if the an existing process is being duplicated.

Midplane and Fusion results

Volumetric shrinkage

When looking at the packing analysis, the result **volumetric shrinkage at ejection** is of primary importance. This shows the distribution of volumetric shrinkage on the part when the part is ejected. This is the primary result that is used to compare one packing result to another. Figure 465 in the left two images shows the volumetric shrinkage of two studies. The top study uses double the packing pressure of the bottom image, as a result the shrinkage is much lower.

The right two images of Figure 465 show the volumetric shrinkage as a path plot. The volumetric shrinkage plot is an intermediate plot so it has multiple results at different times, unlike volumetric shrinkage at ejection which is a single dataset result. The path plot was used to show how the shrinkage varies from the gate to the last place to fill. Because volumetric shrinkage is an intermediate result, the path plot can be animated. This will allow you to see when an area stops shrinking.

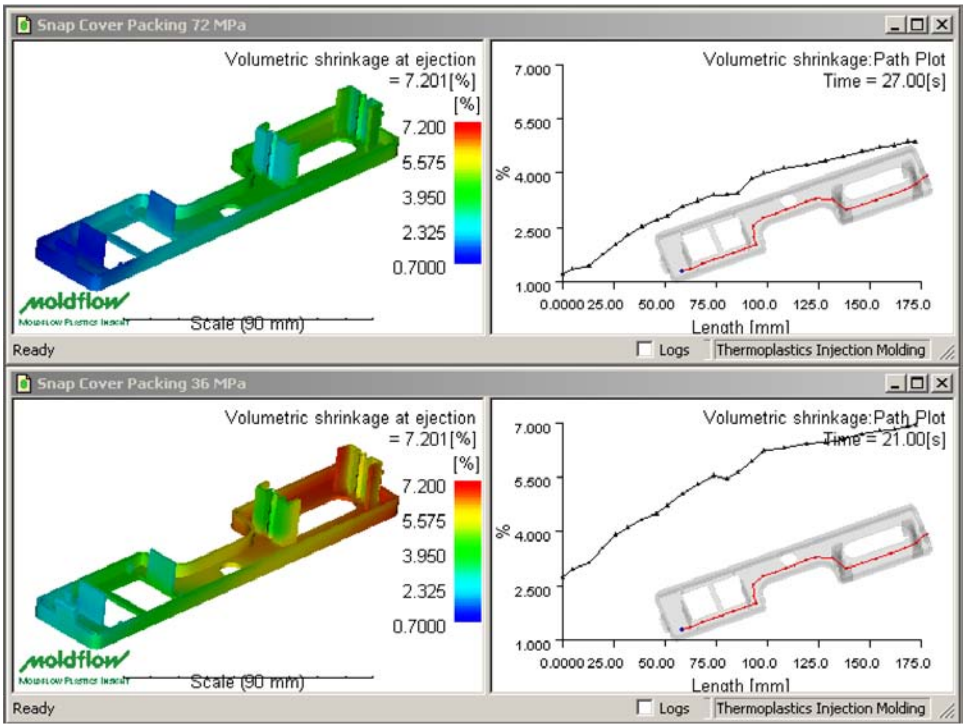


Figure 465: Volumetric shrinkage results

Frozen layer fraction

The frozen layer fraction plot is a useful tool to determine when the gate freezes off. The frozen layer fraction value of 1.0 means that the entire cross-section of the part is at or below the material's transition temperature. Nodal averaging should be turned off when viewing the results to see the best gate freeze time. Because the frozen layer fraction plot is an intermediate result, the gate freeze prediction time will be no more accurate that the difference in the time steps. Figure 466 shows the frozen layer fraction for two packing analyses. The packing pressure in the left result is twice that of the right one. The freeze time in this case, is 9.75 seconds for both parts. The plots below are shown at 9.5 seconds. with a finer resolution with the intermediate results a difference in freeze time may be detected. Change the material, melt temperature, injection time, or packing pressures and the differences could be much greater.

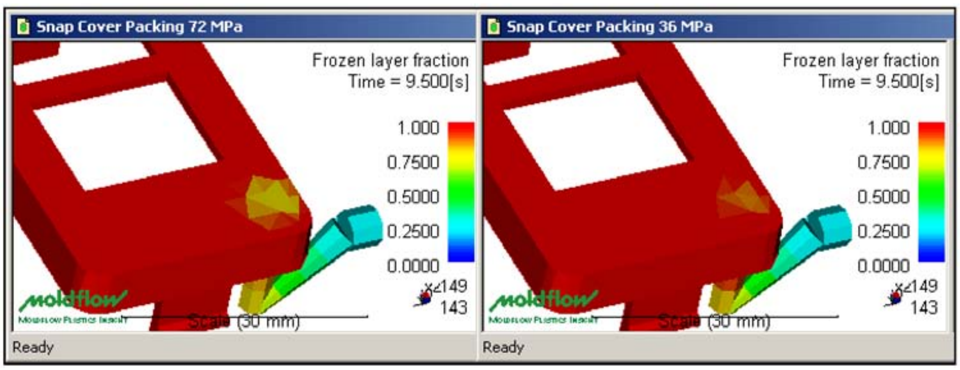


Figure 466: Frozen layer fraction

Pressure XY plot

Generally, the goal of a packing analysis is to achieve a uniform volumetric shrinkage in the part. The pressure on an area of the part when it freezes determines the volumetric shrinkage for that area. The more the pressure applied to the part, the less shrinkage. There is always a pressure gradient in the part so there will be a higher volumetric shrinkage at the end of fill, and lower shrinkage near the gate. Viewing how pressure changes in the part and when areas of the part freeze helps to understand how the shrinkage distribution is formed. Figure 467 shows two packing analyses. The pressure XY plot shows there is a significant pressure difference between the gate and end of fill. This is reason why there is a volumetric shrinkage variation through the part.

It is also clear that the maximum pressure in the part, at any given location, is higher with the packing pressure of 72 MPa compared to a packing pressure of 36 MPa.

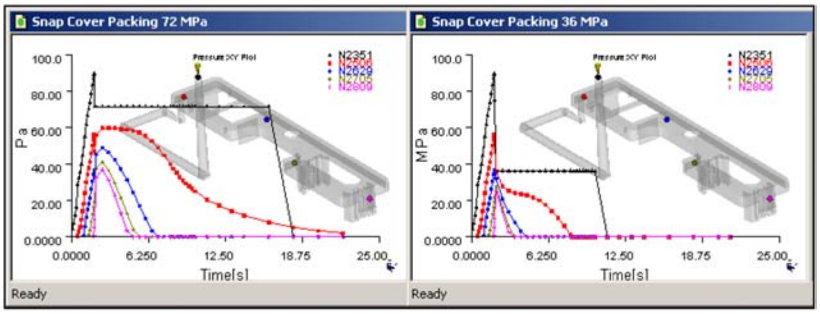


Figure 467: Pressure XY results

Hold pressure

Hold pressure shows the maximum pressure that is applied after V/P switchover. The maximum value is typically the pressure at switchover, assuming the packing pressure is lower than the pressure at switchover. This highest value would be at the injection location, generally at the top of the sprue. The plot is most useful without the feed system so the range of hold or packing pressures can be seen on just the part. The more uniform the pressure is, the more uniform the volumetric shrinkage will be. This plot will highlight areas that a underpacked by having a very low hold pressure compared to the rest of the part.

Figure 468 shows two different packing analyses with the scale for hold pressure set to the same values. The analysis with the highest packing pressure has the highest hold pressures and a slightly smaller difference between the minimum and maximum.

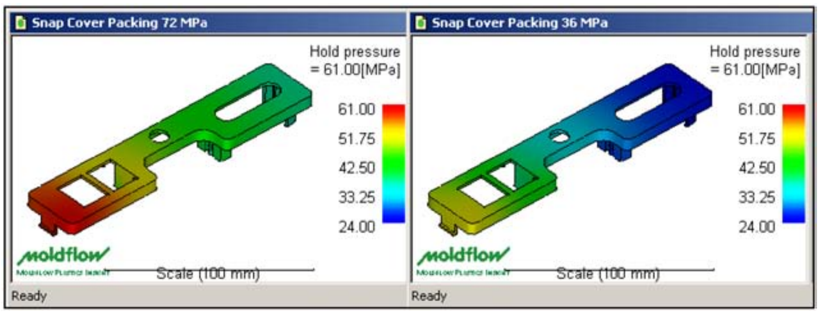


Figure 468: Hold pressure

3D results

Volumetric shrinkage

With a 3D model, the result **Volumetric shrinkage (3D)** is the primary packing result. This result can show us the volumetric shrinkage not only across the part, but through the thickness. There will generally be a greater change in volumetric shrinkage through the thickness than there will be on the surface of the part from the gate to the end of fill.

There are several techniques that can be used to look at the volumetric shrinkage on a 3D part, including:

- Shaded plot with a cutting plane.
- Single contour, set to some value and animated through time.
- Single contour, at ejection (single data set), with a value range.
- Probe plot.

These different techniques are used to look at the results in various ways depending on the objectives of the analysis. Figure 469 shows examples of different ways to look at volumetric shrinkage.

Shaded with cutting plane

Figure 469A shows the volumetric as the default shaded image using a cutting plane to show variation through the thickness. This can be a very good way to look at different sections of the part and get a sense of how volumetric shrinkage changes both through the thickness and across the part.

Single contour with time animation

Figure 469B shows an example of plotting volumetric shrinkage at a specified shrinkage at various times. Specify a single contour plot and a value of volumetric shrinkage of some value. The animation through time shows how the value is distributed through the part. Typically, the area at this shrinkage should get smaller at time progresses.

Single contour with value range animation

Figure 469C shows volumetric shrinkage at ejection (last intermediate result), within a range of shrinkages. The scale of the result should typically be set to round values as with the case in this plot. A single contour is set, and a value range is used that is a fraction of the scale set. In the case of this plot 0.1%. When the result is animated, to will only show areas that are within the range specified. This can be a handy way to look at various locations in the part to see what their shrinkage is compared to the rest of the part. In this plot, there are just a few areas with shrinkage higher than 2.9%, and all of them are in the half of the part furthest from the gate.

Probe plot

The probe plot, as shown in Figure 469D, shows the change of volumetric shrinkage through the part as an XY graph. Generally, the plot is created through the part. Curve 1 is closest to the gate, and Curve 4 is furthest. You can see that the shrinkage on the surface of the part is nearly identical for all of the locations. However, the volumetric shrinkage in the center of the part is a bit higher for curves 3 and 4 which are furthest from the gate.

Summary

In comparing the volumetric shrinkages in the four plots of Figure 469, you can see that you can get the same information in the different plot methods, it is just presented in different ways. You may find that one method is more convenient than another depending on the objectives of the project.

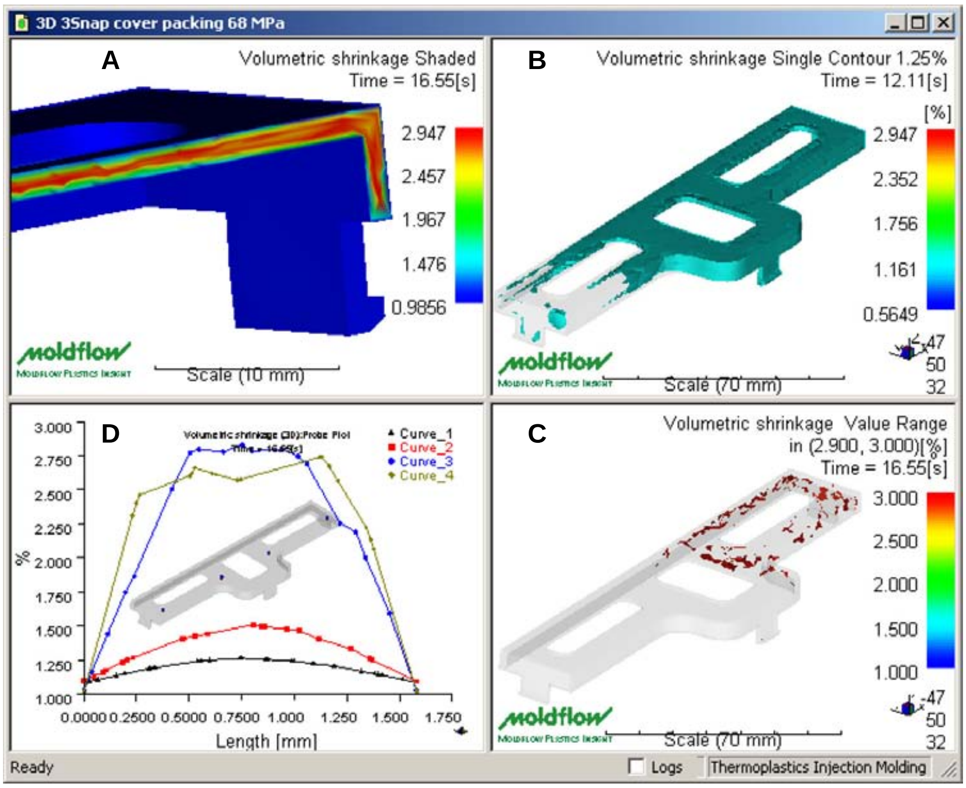


Figure 469: Volumetric shrinkage on a 3D part shown 4 different ways

Temperature 3D

The **Temperature (3D)** result is used to review how the temperature changes over time through the cross section. This is normally done with a shaded plot and a cutting plane. This is a good way to get a general sense for how the part is cooling. The temperature plotted as a single contour is used to determine when the gate is frozen, preventing the cavity from being packed out. This is done by referencing the transition temperature for the polymer.

The gate freeze time accuracy is based on the number of intermediate results. The default number of intermediate results is five. If the packing + cooling time is ten seconds, the gate freeze time is accurate to within 2 seconds. Generally, the number of intermediate results is partly determined by the packing + cooling time and the tolerance desired for the gate freeze.

Two ways of viewing temperature results is commonly done including:

- Shaded, cutting plane, scale all frames.
- Single contour, at transition temperature.

Shaded a cutting plane

The temperature plot with a cutting plane shown in Figure 470A is shown when the part has just filled. The gate is on the left side. The temperature in the center of the cross-section is noticeably hotter on the by the gate. Watching how the cross section cools down can influence how the part is packed out, and therefore the volumetric shrinkage.

Single contour, at transition temperature

The easiest way to determine when the gate is frozen is to look at the temperature as a single contour with the value set to the transition temperature. When the plot separates in the gate, the gate have frozen. This is illustrated in Figure 470B. Animate the result one frame at a time until the gate separates from the part.

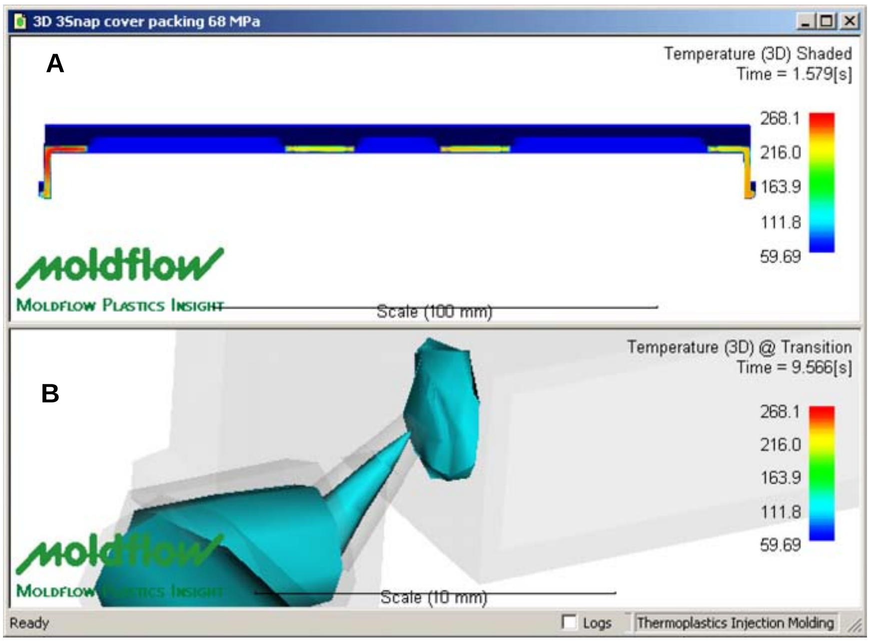


Figure 470: Temperature plots

Pressure XY plot

Generally, the goal of a packing analysis is to achieve a uniform volumetric shrinkage in the part. The pressure on an area of the part when it freezes determines the volumetric shrinkage for that area. The more the pressure applied to the part, the less shrinkage. There is always a pressure gradient in the part so there will be a higher volumetric shrinkage at the end of fill, and lower shrinkage near the gate. Viewing how pressure changes in the part and when areas of the part freeze helps to understand how the shrinkage distribution is formed. Figure 471 shows two packing analyses. The pressure XY plot shows there is a significant pressure difference between the gate and end of fill. This is reason why there is a volumetric shrinkage variation through the part.

It is also clear that the maximum pressure in the part, at any given location, is higher with the packing pressure of 68 MPa compared to a packing pressure of 34 MPa.

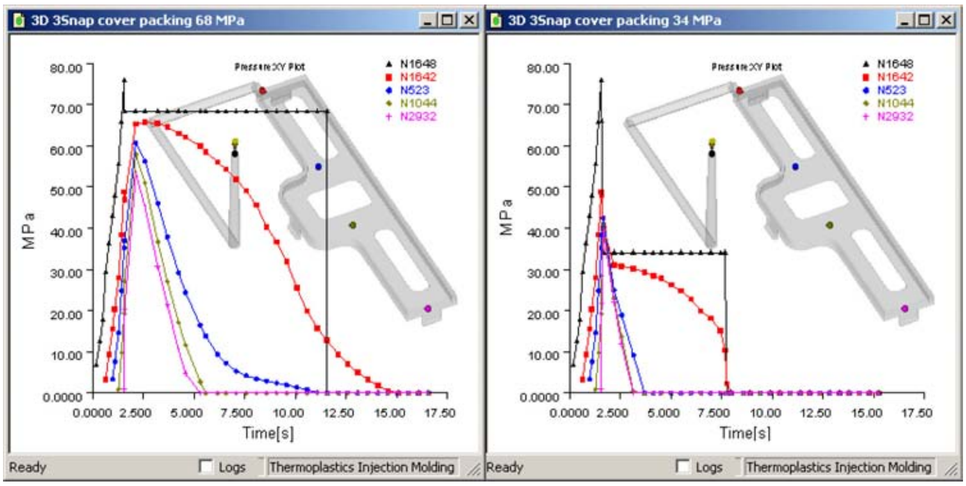


Figure 471: Pressure XY plots on two 3D analyses

What you've Learned

A packing analysis should be done after the filling of the part is optimized, the runners and size/balanced, and preferable after a cooling analysis is done. The gate location, processing conditions, sizes of the gates and runners, and the heat extraction capability of the mold all have an influence on the packing of the part.

Packing pressure and holding pressure can be used interchangeably. It is the amount of pressure applied to the polymer after the Velocity/Pressure switchover. The pressure can be a profile rather than a constant pressure. The packing or holding time is how long the pressure is being applied. Cooling time is determined from when the pressure is released until the mold opens.

To run a packing analysis, the method of switchover, the packing profile and the cooling time must be entered. The switchover and packing profile are needed to run a filling analysis but are not typically changed until the packing is being optimized. The packing time should be low enough so the clamp force limit on the machine is not exceeded and should be high enough to adequately pack out the part. The packing time must be long enough so the gates freeze before pressure is released.

For midplane and Fusion models, the following results are viewed.

- Volumetric shrinkage.
  - This is the primary plot to view. The more uniform the shrinkage, the better.
- Frozen layer fraction.
  - This plot is used to determine when the gate freezes. The frozen layer should be viewed with nodal averaging off. The resolution of the gate freeze is controlled by the number of intermediate results and the packing + cooling time.
- Pressure XY plot.
  - The pressure shown as an XY plot is used to show how the pressure decays in various regions of the part and how the pressure varies between the locations selected.
- Hold Pressure.
  - Hold pressure is another plot that can be used look at the pressure gradient across the part. The more uniform the packing pressure, the better.

For 3D results, the same type of information is required as with midplane and Fusion, but different techniques are used. Volumetric shrinkage results has four ways in which you can plot the results. A shaded image with a cutting plane and using single contours are common methods. The result Temperature 3D is used at the transition temperature to determine the gate freeze with 3D parts. The pressure as an XY plot is used in much the same way in 3D as it is with midplane and Fusion.

CHAPTER 19

Using Valve Gates

Aim

This chapter will introduce you to the modeling and analysis of valve gates.

Why do it

Valve gates are a common form of hot runner component. Valve gates create a positive shut off so the plastic flow front and packing can be controlled by opening and closing the valve gate as required. One common use of valve gates is to eliminate weld lines, which is sometimes referred to as sequential gating. One gate is open and a second gate is not opened until the flow front from the first gate has passed the location of the second. When the second gate is opened, no weld line is formed.

Overview

In this chapter, you are provided with a model of a tub that requires 3 gates. There is a requirement for no weld lines, so valve gates will be used. The runner system will be created using the Runner System Wizard, and then further editing of the runner and gate properties will be required.

The gates will be modified to turn them into valve gates. This requires making the last element in the gate a valve gate controller. Each gate will have a different controller to provide the greatest flexibility. The center gate will have an initial state of open, and the outer two will have a state of flow front.

Theory and Concepts - Using Valve Gates

A valve gate is a gate in a hot runner that is a positive shut off gate by using a pin, as shown in Figure 473. The valve pin opens and closes with assistance from hydraulic or pneumatic cylinders. The valve gate controller can be programmed to open and close a gate as needed in order to control the flow in the cavity. Gates can be opened and closed several times during the filling and packing stages of cycle.

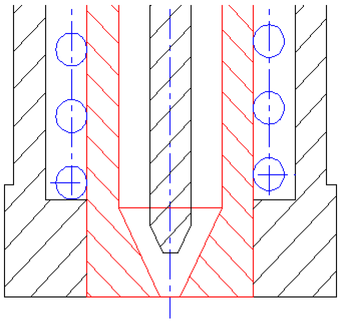


Figure 473: Valve Gate

Valve gates can be used for several reasons, including:

- **Sequential gating**

  With sequential gating, valve gates are used on large parts that can't be filled with one gate, but can't have the weld lines created by two flow fronts meeting. The first gate that is opened will be at one end of the part. When the flow front passes the location of the second gate, the second gate will open. Depending on the situation, the first gate may close or it could stay open. This process will continue until all the gates are open and the part is filled.

- **No gate vestige**

  Sometimes valve gates are used because the vestige left on the part is very difficult to detect. It should look like an ejector pin mark. This may be less objectionable than a typical gate mark left by a sub gate or another type of hot drop tip.

- **Packing control**

  Since valve gates have a positive shut off, the end of packing is controlled by the time when the valve gate is shut. This gives the processor a little more control on the process.

- **Balancing**

  Some family tools use valve gates as a method of balancing. If each part is filled with valve gates, opening and closing the valve gates can achieve a balance. This method of using valve gates is generally NOT recommended. It is not normally the best way to balance.

Modeling valve gates

Valve gates can be used with all three mesh types for the part model. The valve gate itself is a single beam element, attached to the part, with a hot gate property, and has a valve gate controller assigned to the beam element. The hot drop that the gate is in, must be modeled. Depending on the style of hot drop, the valve pin may go through the flow channel or it may not. With most valve gates, the gate pin does go through the center of the flow channel, making the flow channel annular. There are some new styles of valve gates where the valve pin and flow channel down the drop are parallel, and the plastic only gets to the axis of the pin right at the bottom. The element(s) that represent the valve gate itself, have the diameter of the valve pin, with a length equal to the stroke length of the valve pin.

For 3D models, to model valve gates, the runner system must be constructed with beam elements.

Modeling an annular hot drop

To model an annular hot drop, the beam element properties must be set, as shown in Figure 474. The property **hot runner** must be chosen. The cross-section is annular, and in most cases, the shape is non-tapered. The outer diameter is the bore diameter of the drop, and the inner diameter is the diameter of the valve pin. The outer heater should be at the melt temperature unless a specific temperature is known. The inner heater temperature should be the same as the outer temperature.

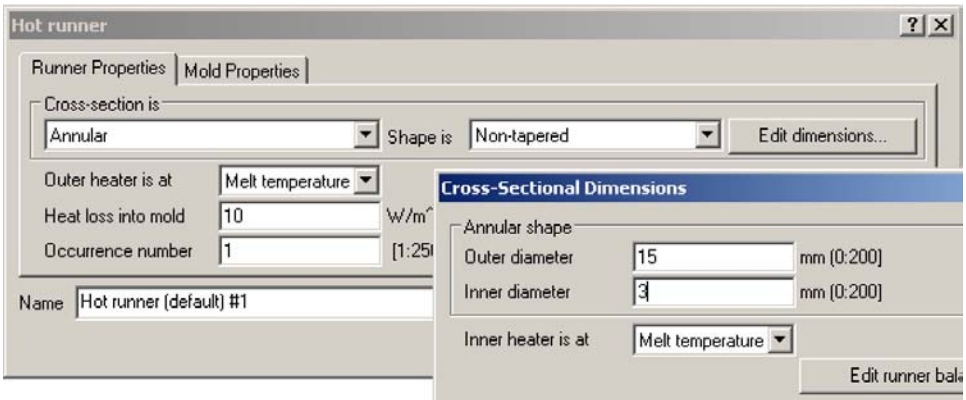


Figure 474: Hot drop (annular cross section) properties

Modeling a circular hot drop

If the hot drop is circular, the beam elements will need to be specified as hot runners, circular and the diameter. The same outer heater considerations should be made for circular hot drops, as it is for annular.

Modeling the valve gate

Adding a valve gate controller to a hot gate property defines a valve gate. The following rules apply:

1. There must be a valve gate controller for every valve gate on the part.
2. For every valve gate that is to be controlled differently, there must be a different valve gate controller.
3. Changing the name of the valve gate controller is the best way to verify each valve gate has the correct controller.

> Assign a different valve gate controller to each gate and use a different name for each controller to distinguish them. This will give you the best flexibility for programming the valve gates.

Depending on how the gate is made, there may be more than one element used to define the gate. Only the last element of the gate should have a controller assigned. The rest of the elements in the gate should have no controller assigned to them.

Valve gate controller assignment

On the **Hot gate** property, **Valve Control** tab, the valve gate controller is selected and edited. Initially, the controller is selected by clicking the **Select** button and picking the only controller in valve gate controller database.

Click the **Edit** button to open the **Valve gate controller** dialog. From here the valve gate control and the name of the controller is set. The default name for the controller should be changed to a unique name that can distinguish it from other gate controllers. Click the **Edit settings** button to set the timing of the controller.

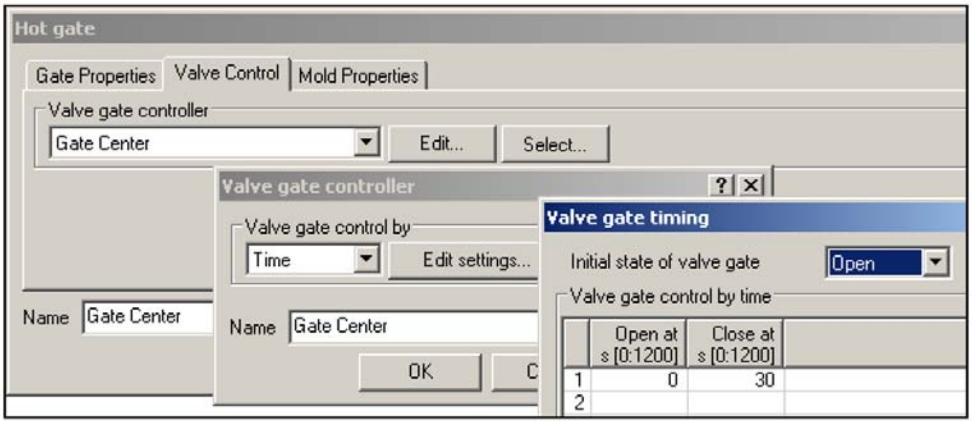


Figure 475: Edit a valve gate

Control methods

Valve gates have several methods of control. These methods are listed in Table 82. Each control method has a different combination of parameters that it can support. A description of the control type is in Table 83 and a description of the parameter is in Table 84.

Table 82: Valve Gate Control and Parameter Matrix

Control Type / Parameter	Time	Flow Front	Pressure	% Volume	Ram Position
Initial State	✓		✓	✓	✓
Trigger Location		✓	✓		
Delay Time		✓			

Table 83: Valve Gate Control Types

Control Type	Description
Time	Allows you to enter time values, measured from the start of injection, at which the valve gate will be opened and closed. This is the most commonly used method.
Flow front	Specifies the valve gate to open when the flow front in the part reaches a specified location and then open/close at specified times thereafter. This is used to simulate and setup sequential gating. At least one valve gate must have time as the control, and open as the initial state.
Pressure	Specifies the valve gate to open/close when the pressure reaches a specified level at a specified location and then open/close at specified pressures thereafter.
% Volume	Specifies the valve gate to open/close when a specified percentage of the cavity has filled and then open/close at specified volumes thereafter.
Ram position	Specifies the valve gate to open/close when a specified ram displacement is reached and then open/close at specified ram displacements thereafter.

Table 84: Valve Gate Parameters

Parameter	Description
Initial State – Open	Select this option if the valve gate is initially open and the first event you will specify is the closing of the valve gate.
Initial State – Closed	Select this option if the valve gate is initially closed and the first event you will specify is the opening of the valve gate.
Trigger location – Gate	Select this option if you want the initial opening/closing of the valve gate to be triggered by an event at the gate node associated with the selected valve gate.
Trigger location – Specified	Select this option if you want the initial opening/closing of the valve gate to be triggered by an event at a specified node in the model.
Node No.	Specifies the node for the Trigger location.
Delay time	Specifies that the valve gate will be opened at the required time, in seconds, after the flow front has reached the trigger location. If you do not want a delay time to apply, enter **0**.

Control valve gate timing

By property

Valve gates can be opened and closed several times during the cycle. In most cases, the opening and closing is done by time. Figure 476 shows an example of a valve gate that is controlled by time, with an initial state of open, but the gate is closed once then opened again before closing. The changing of the valve gate state can be during fill or pack.

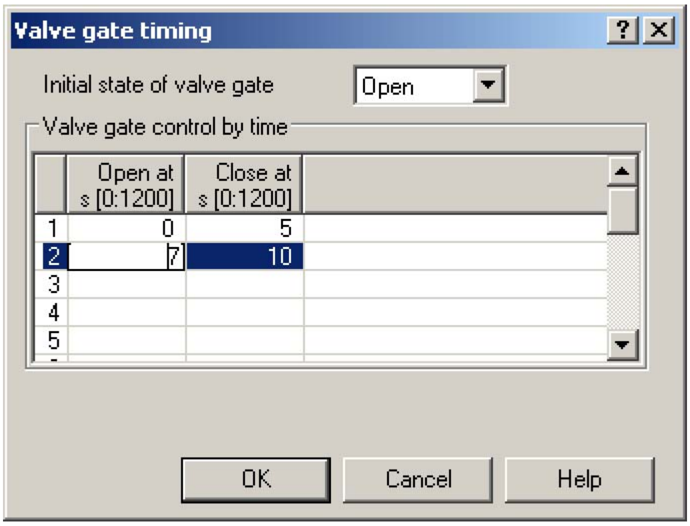


Figure 476: Valve gate timing by the controller property

By valve gate timings dialog

Once the valve gate properties have the correct control type, a dialog, shown in Figure 477, can be opened by **Analysis** ➡ **Edit Valve Gate Timings** to change the times used to open and close the valve gates. Pick the control method of the gates you want to change. All the gates for that control method will be listed. Modify the parameters as necessary and click OK to exit the dialog.

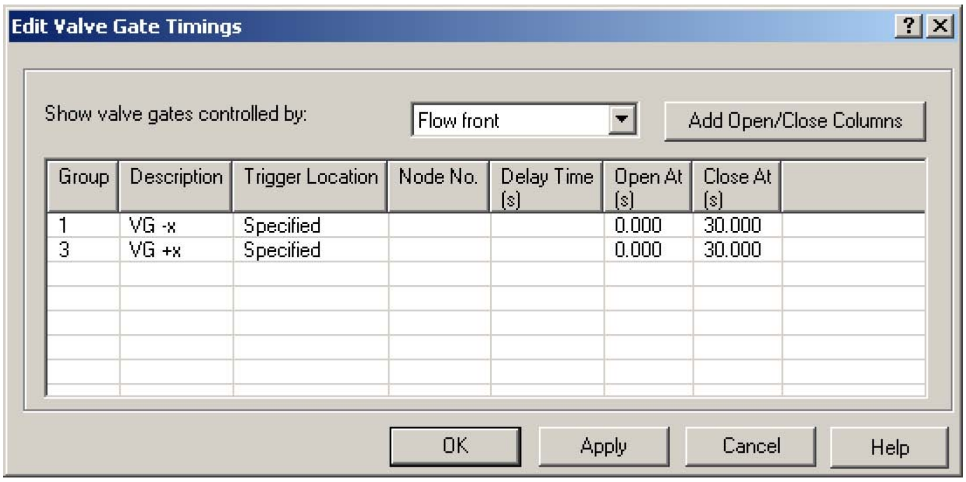


Figure 477: Valve gate timing by valve gate timings dialog

What You've Learned

Valve gates can be applied to all mesh types.

Valve gates are a type of gate which can be opened or closed by a controller, and allow extra control over the filling of a part.

Valve gates may be used for several reasons including:

- Use sequential gating to avoid weld lines.
- Reduce gate marks.
- Have more control over packing.
- Balance the flow in family molds.

Valve gates are modeled with hot gate beam properties. A valve gate controller must be assigned. The control type must be set to one of the following:

- Time.
- Flow front.
- Pressure.
- % Volume.
- Ram position.

The parameters for the control types include:

- Initial State – Open.
- Initial State – Closed.
- Trigger location – Gate.
- Trigger location – Specified.
- Node No.
- Delay time.

CHAPTER 20

Flow Leaders and Deflectors

Aim

The aim of this chapter is to use a flow deflector to move the location of a weld line.

Why do it

Flow leaders and deflectors are small changes in wall thickness to accomplish a balance or to move a weld line.

Flow leaders and flow deflectors can be useful tools to solve balance problems within parts or to move weld lines.

Design constraints sometimes prevent us from moving gates. For example, a particular part may have a shape that doesn't allow for a gate location that creates a balance in the part with a nominal wall thickness. In cases like this, flow leaders and deflectors can be used to move weld lines.

Overview

In this chapter, you will move the weld line around a square hole from the center of the hole's edge to a corner. You will achieve this by creating changing wall thickness to form a flow deflector in the relevant section of the model.

Moving the weld line makes the weld line less visible to the user because of its location and the angle at which it is formed.

In this practice, you will perform the following:

- Run a filling analysis.
- Review filling results.
- Review the weld line location.
- Create the flow deflector by decreasing the wall thickness.
- Re-run and review results.

Theory and Concepts - Flow Leaders and Deflectors

What are flow leaders and deflectors?

- **Flow leader**
  - A local increase in thickness from the nominal wall of the part, as shown in Figure 486. Material flows more easily in the thicker wall so the material is lead in the thicker area.
- **Flow deflector**
  - A local decrease in wall thickness from the nominal wall of the part, as shown in Figure 486. Because the wall is thinner, material hesitates slightly and has preferential flow down the nominal wall or some thicker area.

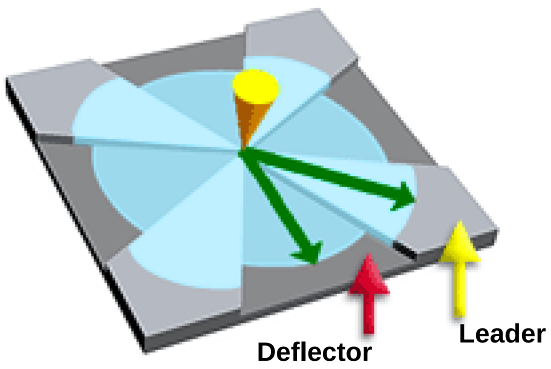


Figure 486: Flow leaders and flow deflectors in a square plate part

How flow leaders and deflectors are used

Both flow leaders and deflectors do the same thing, they control the shape of the flow front. Flow leaders and deflectors are used to balance flow paths in the part or to move weld lines. By creating a balanced fill in the part the warpage due to orientation effects is reduced. Many people refer to flow leaders and deflectors as flow leaders even though the thickness may get smaller.

Advantages and disadvantages

Both flow leaders and deflectors have advantages and disadvantages:

Table 86: Advantages and disadvantages of flow leaders and deflectors

	Advantages	Disadvantages
Flow leaders	Reduce the shear stress level in the part. If flow leaders were not initially incorporated into the part design, flow leaders are steel safe because they are created by, removing steel from the mold.	Flow leaders add material to the part. Potentially the cycle time is increased if the flow leader is the thickest area on the part.
Flow deflectors	Flow deflectors reduce the wall thickness saving material and possibly cycle time.	Potentially the structural integrity could be compromised so care must be taken if wall thicknesses are reduced to balance the flow or move a weld line.

Designing the thickness change

The change in wall thickness for making a flow leader or deflector should be small. The change should be under 25% of the nominal wall. This is not always possible.

The thickness change in a flow leader should be under 25%

A flow leader or deflector should have a gradual change in thickness. In the plate shown in Figure 486, the thickness changes are abrupt to show flow leaders clearly. In real parts, the best transition is a constantly changing wall, shown in Figure 487. In the past, to lead the flow in the part a deep narrow groove was machined into the part. This is a poor practice as it creates more of a river effect and may be difficult to pack the part to eliminate sink marks caused by the deep groove.

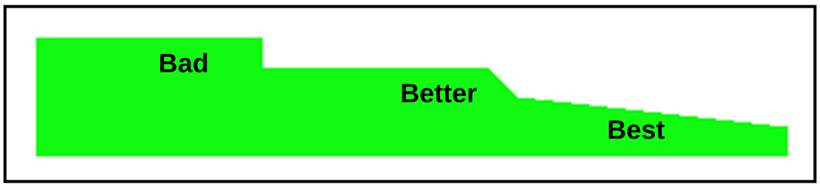


Figure 487: Wall thickness transitions

Creating a flow leader

To simulate a flow leader or deflector, you need to select elements and change their thicknesses.

In Figure 488, the elements on the top left side of the model were created specifically for adding a flow deflector. The boundaries of the deflector were carefully modeled. When you plan ahead carefully like this, you can model the location of the flow leader in the CAD system and translate the model into MPI with the area already defined. The elements of the flow deflector were put on a separate layer for easy identification.

If you decide to add flow leaders or deflectors during the analysis process, you can run some initial analyses by just selecting the elements as they currently exist. Later, you can improve the model by defining the edges of the flow leader more clearly (as on the top half of the figure below). If possible, model the flow leader on a separate layer. This makes it easier to select the elements needed to adjust the thickness later.

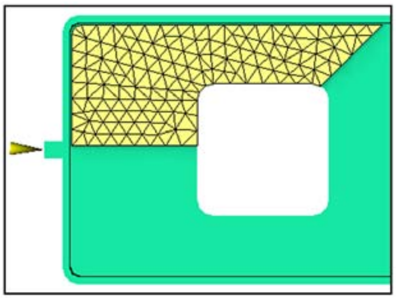


Figure 488: Mesh for a flow leader

Midplane models

When you are simulating flow leaders with a midplane model, it is quite easy to change the thickness by selecting the elements that make the flow leader and create a new part surface property with the new thickness, as shown in Figure 489. A midplane analysis can really only calculate the step transition (the "bad" transition shown in Figure 487). This influences the results slightly. Even though the transition can only be a step in the analysis, the tool should at very least have a chamfer at the thickness transition.

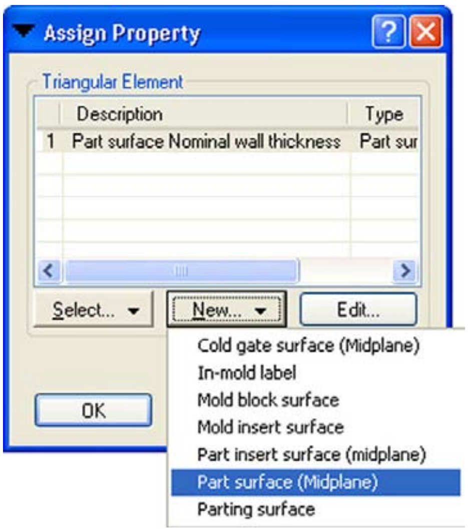


Figure 489: Assign property dialog

Fusion models

For Fusion models, you can start the design of flow leaders in the same way as you would for a midplane model. That is, you can select elements and then change their thicknesses. When doing this on a Fusion model, you should select elements on both sides of the wall thickness.

As with midplane, the initial boundary can be jagged because of the mesh density, but once the flow leader size has been established, you can straighten the boundary. You can do this with the Align Nodes mesh tool in MPI, or you can do it on the CAD model and retranslate it.

Because with Fusion, the chamfer or continuing taper can be modeled it is recommended that when the flow leader thickness in known, the part model should be retranslated from the CAD system with the finish flow leader in it for a final verification.

Also, if the analysis with this part will go past flow, you should model the flow leader in the CAD system and then retranslate the Fusion model. In cool and warp analyses, the correct geometry must be present, not just a property defining the thickness.

> If that analysis is going into cooling and warpage, the thickness representing the flow leader can't be manually set. The thickness property of the elements must be the same as the elemental spacing.
>
> Once the flow leader/deflector is sized, make the changes in the cad model and re-import before continuing on to Cool and Warp.

3D models

Since 3D models have a true volume mesh, there is no thickness assignment. You can test the flow leader/deflector as a Fusion model then confirm the changes with a 3D model. This would require building the leader/deflector in CAD and importing it back into MPI.

What You've Learned

- Flow leaders are subtle increases in thickness.
- Flow deflectors are subtle decreases in thickness.
- Both are primarily used to help balance the filling of a part or reposition weld lines.
- The change in wall thickness should be less than 25% of the nominal wall.
- Flow leader advantages include:
  - Reduce shear stress levels.
  - Is steel safe as steel is removed to create a leader. The can be designed after the tool is built.
- Flow leader disadvantages include:
  - Volume is added to the part increasing the cost.
  - Potentially, the cycle time will increase.
- Flow deflector advantages include:
  - Reduce material volume therefore material costs.
- Flow leader disadvantages include:
  - Possible structural integrety problems.
- Flow leaders/deflectors should be made with gradual changes in thickness not abrupt changes.
- For midplane and Fusion models, element properties are modifed to change the thickness of the part. Using layers is a convienint way to manage the elements used to form the layers.
- For Fusion models, if analysis work is going to go past a flow analysis, the flow leader should be modeled in the CAD system and re-imported, and then verified.
- For 3D models, the elements have no thickness properties. Flow leaders can be sized using Fusion and verified by 3D, or for each iteration, a new CAD model much be created and imported.

CHAPTER 21

Flow Analysis Process Settings

Aim

The aim of this chapter is to review all of the options and settings available for a flow analysis.

Why do it

It is important to know and understand all of the options and settings available for running a flow analysis. Understanding the options for an analysis will allow you to better change the options if needed to investigate a problem.

Overview

This chapter discusses all mesh types. Many of the settings are the same between the mesh types. There are cases where the process settings are different between the meshes. Most of the differences are with 3D meshes.

This chapter reviews the options and settings available primarily for a flow analysis. There is some discussion of flow analysis variants such as core deflection.

Theory and Concepts - Flow Analysis Process Settings

Most settings and parameters are the same between the 3 mesh types below is that describes the similarities and differences in the settings for each mesh type.

Table 87: Process settings for the mesh types

Similarities	Differences
Flow settings	• Solver parameters Different tabs and parameters
Advanced options	
• Molding material properties	
• Process controller	
• Injection molding machine	
• Mold material	

Flow settings dialog

In the **Process Settings** wizard, there is one page for a flow analysis, shown in Figure 494. The most common settings for a flow analysis are done on this page. Depending on the setting selected, you may need to open dialogs accessible from this main page. The fields and dialogs available from the flow settings page include:

- Mold surface temperature.
- Melt temperature.
- Filling control.
- Velocity/pressure switch-over.
- Pack/holding control.
- Cooling time.
- Advanced options.
- Fiber Parameters.

Each of these will be described in detail.

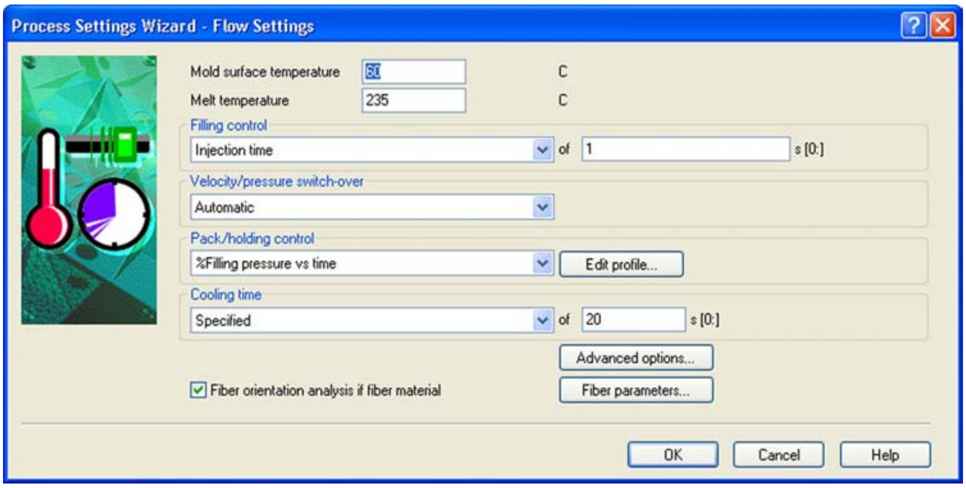


Figure 494: Flow settings

Mold surface temperature

This is the plastic/metal interface or the surface of the mold that the plastic hits when injected into the mold. The default value comes from the material database and is a mid range temperature as recommended by the material manufacturer. The mold surface temperature is often changed based on a molding window analysis or if you wanted to use a different value. The mold temperature must **NOT be higher than the ejection temperature** of the material.

If the analysis sequence is Cool + Flow, the mold surface temperature will not be on this page, as it was defined on the cooling process settings page.

Melt temperature

This is the temperature of the polymer as it enters the model at the injection location(s). The default value comes from the material database and is a mid range value. The melt temperature **must be above the transition temperature** of the polymer. Often the melt temperature is adjusted based on a molding window analysis or if you wanted to use a different value. The melt temperature can have a significant effect on the quality and mold ability of the part.

If the analysis sequence is Cool + Flow, the melt temperature will not be on this page, as it was defined on the cooling process settings page.

Filling control

The filling control defines how the ram velocity is controlled to fill the part. There are four options that include:

- Automatic.
- Injection time.
- Flow rate.
- Ram Speed Profile.

The first three control methods all use a constant volumetric flow rate. The difference is how that flow rate is defined.

Automatic

The **Automatic** method determines an appropriate injection time/flow rate based on a quick strip analysis similar to the molding window analysis. The automatic analysis uses the mold surface temperature and melt temperature values entered, then calculates an injection time that has a small temperature drop in the flow front at the end of fill. The automatic analysis should only be used on parts without runner systems. The algorithm used to calculate the fill time will calculate an injection time that is too long if runners are part of the model because of the shear heat runners generate.

> The automatic analysis should only be used on parts without runner systems.

Injection time

The injection time is the most common filling control. This defines the time required to fill the part at a constant velocity. The actual fill time will be higher than the value entered primarily due to the compressibility of the polymer. The injection time also assumes the switch-over is at 100% when calculating the volumetric flow rate to be used.

Flow rate

The flow rate option lets you to specify the flow rate directly. This option should be **used when analyzing parts with cold runners**. The optimum injection time for the part should be optimized without a feed system. When runners are added and the same injection time is used, the parts will fill too fast. Calculate the flow rate based on the part volume and the number of parts. When the analysis runs, the fill time will be longer to account for the time required to fill the runners.

Ram speed profile

The ram speed profile have the following options for controlling the filling of the part:

- %Shot volume vs. %flow rate.
- %Stroke vs. %ram speed.
- Stroke vs. ram speed.
- Stroke vs. %maximum ram speed.
- Stroke vs. flow rate.
- Stroke vs. %maximum flow rate.
- Ram speed vs. time.
- %Maximum ram speed vs. time.
- Flow rate vs. time.
- %Maximum flow rate vs. time.

These options are not often used in the initial design stages of a part unless results warrant it. Often the ram speed profile is adjusted to the same settings as a press to help trouble shoot a problem. The many options available for setting the profile reflect the many ways machines can set up a profile, depending on the controller.

For all of these profiles but the first, the machine screw diameter must be set. The default molding machine has no machine screw diameter value. A specific molding machine needs to be specified, or the default molding machine needs to be modified within the study to enter in the machine screw diameter.

%Shot volume vs. %flow rate

This is a generic method of setting a profile. This method is most often used as a design tool to test out the effectiveness of a profile to solve an issue. To use the profile, set the % shot volume and the corresponding % Maximum flow rate. To use this method, a reference flow rate or injection time and a stroke volume must be set. The default is stroke volume is automatic. If the stroke volume is specified, the machine screw diameter and stroke length must be entered. The stroke volume must be larger than the part to prevent short shots.

%Stroke vs. %ram speed

In this profile, after the %stroke length and %ram speed are entered, the reference time or flow rate need to be entered. The stroke volume is required also. As before, the automatic option can be used.

Stroke vs. ram speed

For this profile, that actual position of the ram (stroke) is entered in a liner unit millimeters or inches. The ram speed is entered in a velocity mm/sec. or in/sec. The cushion and shot size must be also entered. The best method to enter a V/P switchover is by volume. Calculate the volume displacement by the stroke to determine the volume to set for the switchover.

Figure 495A shows the dialog for setting up a stroke vs. ram speed profile and Figure 495B is the graph of what the profile looks like. Click the **Plot Profile** button to open the graph.

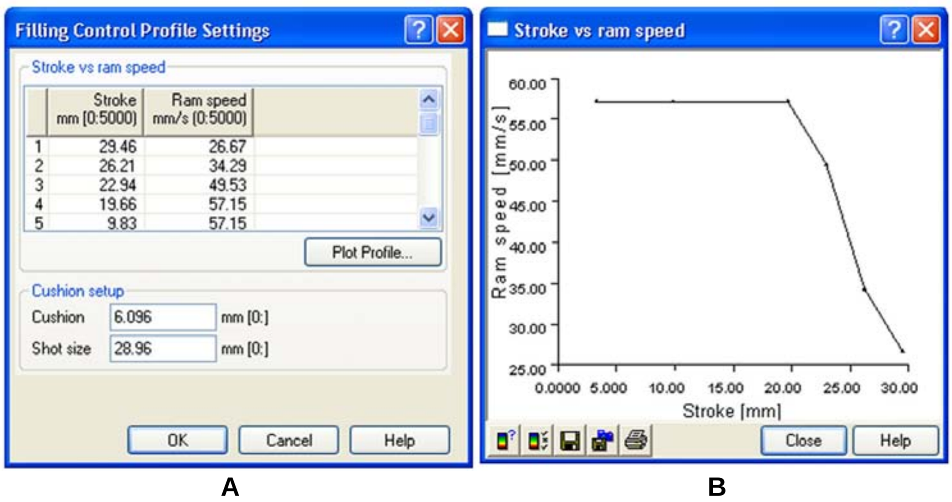


Figure 495: Stroke vs. Ram speed profile

Stroke vs. %maximum ram speed

In this profile method, the %maximum ram speed is entered. The maximum ram speed is the maximum speed the machine is capable of producing. The cushion and shot size need to be entered.

Stroke vs. flow rate

In this profile, the stroke in linear units and the volumetric flow rate are entered. In addition the shot size and cushion are entered.

Stroke vs. %maximum flow rate

In this profile, the flow rate is entered as a percentage of the maximum volumetric flow rate the machine is capable of producing. The stroke is entered in a linear dimension. The cushion and shot size must be entered.

Ram speed vs. time

In this profile, the time from the start of the cycle is entered for every different ram speed. The cushion and shot size need to be entered.

%Maximum ram speed vs. time

With this profile, the ram speed is expressed as a percentage of the maximum of the machine capability, rather than an actual value. This time for each step is measured from the start of the cycle. The cushion and shot size need to be entered.

Flow rate vs. time

In this profile, the time from the start of the cycle and volumetric flow rate are entered. The shot size and cushion are once again entered.

%Maximum flow rate vs. time

With this profile, the flow rate is expressed as a percentage of the maximum of the machine capacity. As in most of the other profiles the cushion and shot size need to be entered.

Velocity/pressure switch-over

There are several methods used to switch the process from a velocity controlled filling to pressure controlled packing. Once the process has switched to pressure control, the rest of the part is filled out. If or when the analysis continues into packing, the profile is continued. The switch-over methods include:

- Automatic.
- %Volume filled.
- Injection pressure.
- Hydraulic pressure.
- Clamp force.
- Pressure control point.
- Injection time.

- Whichever comes first.

Automatic

This option automatically determines the optimum time to switch from velocity to pressure control. The transition point is selected such that if the ram stopped instantaneously, there would be enough melt decompression to just fill the cavity. This is the default switch-over method and is most often used.

%Volume filled

The % volume filled method switches-over from filling to packing when a particular percentage of the cavity volume is filled. By default, this percentage is 99%. This method is typically used for switching over when the automatic method is not used. Sometimes the automatic method switches too early or late and you want control of when the part switches over. It is also commonly used when a specific ram speed profile is used.

Injection pressure

Using injection pressure, the switch-over will occur when the machine reaches a specified injection pressure. The injection pressure is the pressure applied directly to the polymer by the ram (screw) of the machine.

Hydraulic pressure

Hydraulic pressure specifies that the switch-over from filling to packing will take place when the machine reaches a specified hydraulic pressure. The difference between hydraulic pressure and injection pressure is the intensification ratio. The intensification ratio is the difference in area of the back of the hydraulic cylinder that moves the machine's ram and the ram cross-sectional area. The value of the intensification ratio is stored in the injection molding machine database. The default molding machine's intensification ratio is 10:1. Many molding machines have higher ratios, increasing the injection pressure capacity.

Clamp force

Clamp force specifies that the switch-over will occur when the clamp force reaches a specified limit.

Pressure control point

This method switches-over from filling to packing when a specified pressure is reached at a specified location on the mesh. The node in the part and the pressure required at that node are specified in the Process Settings wizard. The node number at the location at switch-over can be determined with the query entity command or by turning on labels for the layer the nodes are on. This is used to simulate using a pressure transducer in the mold for determining switchover.

Injection time

The injection time switch-over specifies that the switch-over from filling to packing will take place at the specified time from the beginning of the cycle. This is not a recommended method of switching over. However, most molding machines will allow more than one method to be specified. This is often a backup method that is used.

Whichever comes first

Select this option if you want to specify one or more of the switch-over criteria listed above. In this case, velocity/pressure switch-over will occur as soon as one of the set criteria is met.

Pack/holding control

There are four methods that can be used to specify the control for the packing phase including:

- % Filling pressure vs. time.
- Packing pressure vs. time.
- Hydraulic pressure vs. time.
- %maximum machine pressure vs. time.

% Filling pressure vs. time

This is the default method of setting the packing profile. The default values are 80% of the fill pressure for 10 seconds. These default values are reasonable starting points most of the time. Once a packing analysis has been done and the results evaluated, the values can be changed if you do not have a specific profile to enter. Machine controllers do not use this method to switch over, but it is a good design method. Figure 4 shows the default profile. The time is entered as a duration, rather than the time from the start of the cycle. The profile also supports a decayed profile. For a given line of the profile the time duration indicates the time required to get to the pressure on that same line from the pressure of the previous line or the V/P switchover pressure.

Once the packing pressure is known the packing pressure vs. time method is often used instead of % filling pressure.

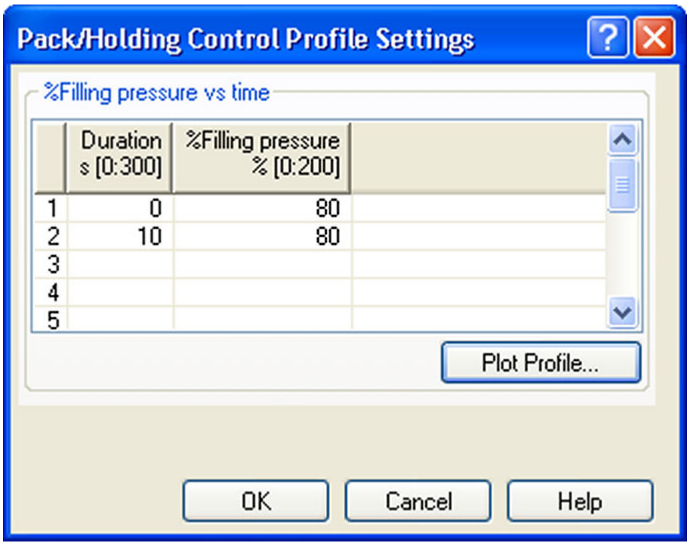


Figure 496: Packing profile

Packing pressure vs. time

If the default %filling pressure vs. time is not used, this is generally the method used. The packing pressure is the injection pressure or the pressure applied directly to the polymer.

Hydraulic pressure vs. time

Sometimes when duplicating an existing process, hydraulic pressure vs. time is used. This assumes the correct intensification ratio is set up.

%maximum machine pressure vs. time

Some machine controllers set the packing pressure by a percentage of maximum machine capacity. This could be used if you are duplicating one of those machines. Using this method requires the actual machine pressure capacity is known and the machine database uses this method.

Cooling time

If a flow analysis is being run without cooling as an input, cooling time must be entered. The default is to explicitly specify a cooling time, and the default value is 20 seconds. The cooling time can be automatically calculated also. When using this option, the ejection temperature and frozen percentage at ejection need to be specified. The defaults here are the ejection temperature of the material as specified in the database and the part must be 100 percent frozen.

Advanced options

For a flow or fill analysis, the advanced options, shown in Figure 497, are accessed with the **Advanced Options** button on the **Flow Setting** page of the Process Settings Wizard. have five categories including:

- Molding material.
- Process controller.
- Injection molding machine.
- Mold material.
- Solver parameters.

For each category, there are an **Edit** and **Select** buttons. The **Edit** button allows you to edit the values of the current item; the **Select** button allows you to choose a new item from a database. Normally, the **Edit** button is used to modify the current settings. The advanced options are the same for all 3 mesh types except for solver options, there are some mesh specific parameters.

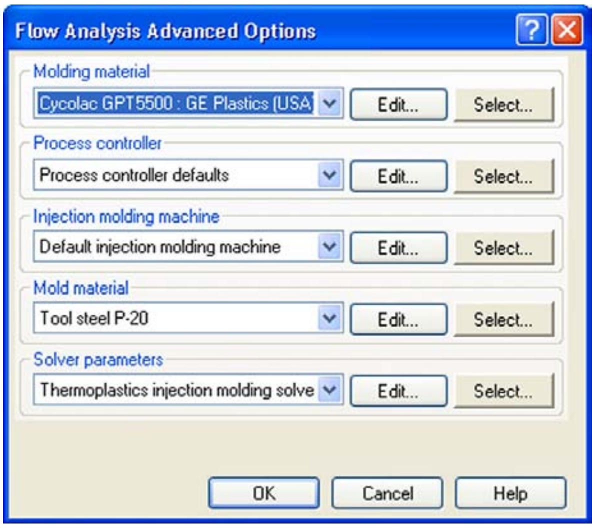


Figure 497: Advanced options

Molding material

The **Select** button for molding material is the same selection dialog that is used when you choose a material from the studies task pane. The **Edit** button opens a dialog that allows you to edit any and all the material properties. It is not advisable to edit the properties for the material. However, sometimes changing the viscosity or shrinkage models are done, as shown in Figure 498. The default viscosity model is Cross-WLF for most materials. A very few materials use Second order as the default model, if the Cross-WLF model is not a good fit. The matrix model can be used with LCP's to better represent their behavior.

For shrinkage models, the default model is the most accurate for the material with the material data available. The Uncorrected residual stress model does not require shrinkage data so if shrinkage testing was not done for a material this is the only model available. If shrinkage data is available, then the CRIMS model is normally the default shrinkage model.

For 3D meshes, a different shrinkage model is used. It is called the Generalized shrinkage model. This is the only one available, so the selection of shrinkage models has no influence for 3D models.

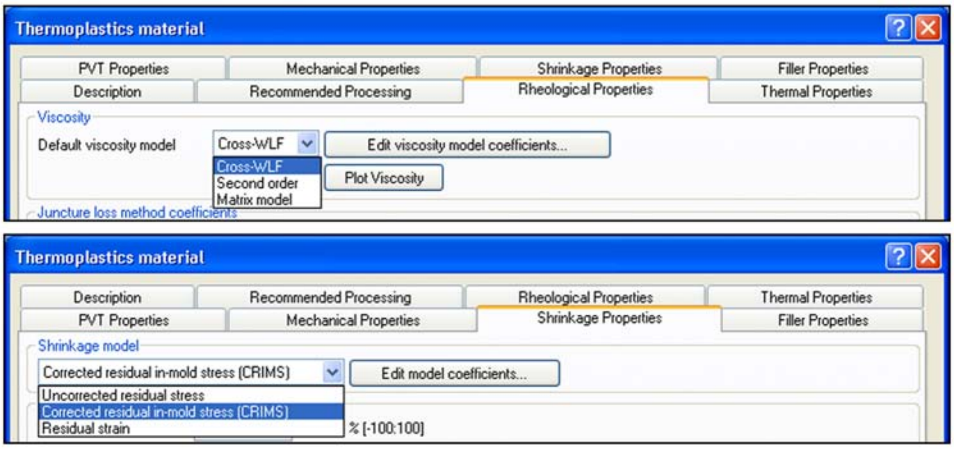


Figure 498: Thermoplastic materials on rheological and shrinkage properties tabs

Process controller

There is only one process controller in a database, so the Select button would only allow you to choose the same process controller again. However, using the **Edit** button, you may want to change some values. The tabs for all 3 mesh types are the same, shown in Figure 499. The tabs are include:

- Profile/Switch-over Control tab.
- Temperature control tab.
- MMS Profile Data tab.
- Time control (fill) tab.

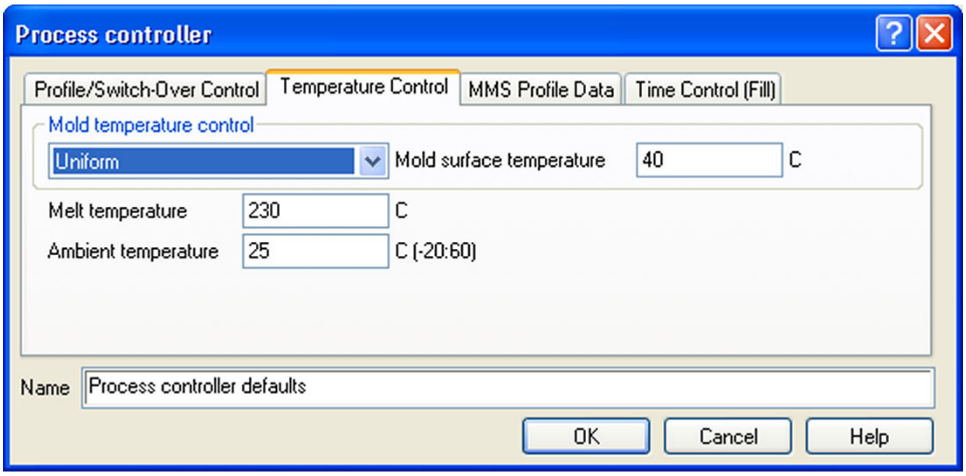


Figure 499: Process controller settings

Profile/Switch-over Control tab

This tab contains the same controls that are on the process settings tab including:

- Filling control.
- Velocity/pressure switch-over.

- Pack/holding control.

These controls are exactly the same settings as the main Flow process settings tab. You will not need to adjust the parameters from this tab.

Temperature control tab

This tab, shown in Figure 499, includes:

- Mold temperature.
- Melt temperature.
- Ambient temperature.

The melt and mold temperatures are the same as the values that on the Process Settings wizard. For mold temperature, there is an option to set a different mold temperature on the cavity and core. To use this option, the properties for the elements must be set. By default, all the elements are given the assignment positive side equals the cavity. Elements can be changed so the positive side equals the core. On midplane models, the negative side can be set also. Once elements have been identified as being cavity or core, the temperatures can be set to different values. This would be a very quick way to look at the effect of how different mold temperatures affect the fill and pack of the part. The best way to do this however is to use a cooling analysis as an input into the flow.

MMS profile data tab

This tab allows you view and edit the velocity and packing profiles that imported from MMS. Profiles are imported with the command **Analysis** ➡ **Import Process data form MPX/Shotscope** ➡ Import process settings.

Time control (fill) tab

This tab allows you to change the mold open time. The default is 5 seconds.

Injection molding machine

An injection-molding machine must be specified so pressure and clamp tonnage capacities can be defined. Depending on other settings being used, other parameters need to be set such as the barrel diameter. The molding machine used by default is called the **Default molding machine**. There is an extensive database of defined molding machines and a few generic machines. If you know the specific machine being used, and you are interested in doing some analysis work that may result into some parameters being close to the limit of the machine, the specific machine should be used. If the specific machine is not listed, the default machine description can be modified, or one of the other machines on the database can be used. Tabs on the dialog for the molding machine include:

- Description.
- Injection unit.
- Hydraulic unit.
- Clamping unit.

Injection unit

The injection unit tab is shown in Figure 500. If a specific injection profile is used, **Machine screw diameter** and **maximum injection stroke** parameters must be specified. The default molding machine does not have any values.

The type of filling control, the rams speed control steps, and pressure control steps are all definable. The control steps default to linear, and can be switched to constant.

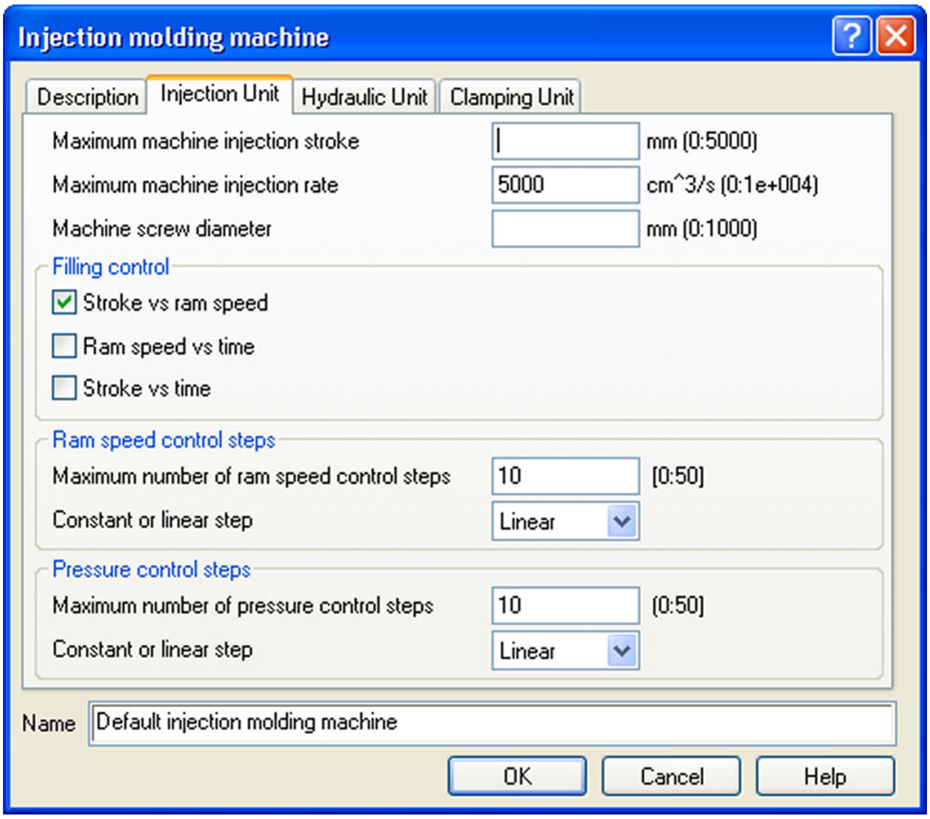


Figure 500: Injection molding machine dialog, injection unit tab

Hydraulic unit

The **Hydraulic unit** tab, shown inFigure 501, can be used to specify the machine's pressure limit and intensification ratio. The pressure limit can be done in hydraulic pressure or injection pressure. The default limit is 180 MPa (~26,000 psi). This pressure is higher than many molding machines. If you are not sure what the machine limit is, you could lower the limit to 140 MPa (20,000 psi). This is not a requirement. However, if the pressure limit is reached, the flow analysis will reduce the flow rate to maintain the pressure. The setting of the pressure will only influence a fill/flow analysis if a limit is reached. The zone plot for a molding window analysis will also be influenced by this setting. Normally for the molding window analysis, the pressure capacity is set to 140 MPa or the press capacity if known.

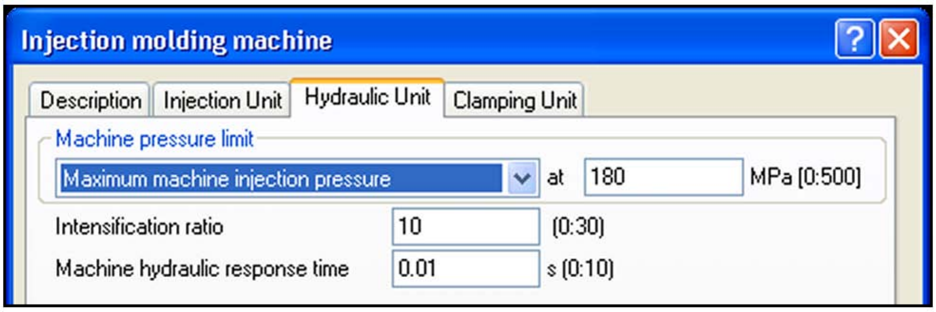


Figure 501: Injection molding machine dialog, hydraulic unit tab

Clamping unit

The **Clamping unit** tab, shown in Figure 502, defines the clamp force capacity of the machine. The default molding machine's limit is 7000 tonnes. This is much higher than most presses, and is used to ensure the limit will not be reached unless the limit is lowered. Also, there is a checkbox indicating the clamp tonnage limit is not to be exceeded. This should be checked so the flow solver will change the flow rate if necessary to maintain the clamp force.

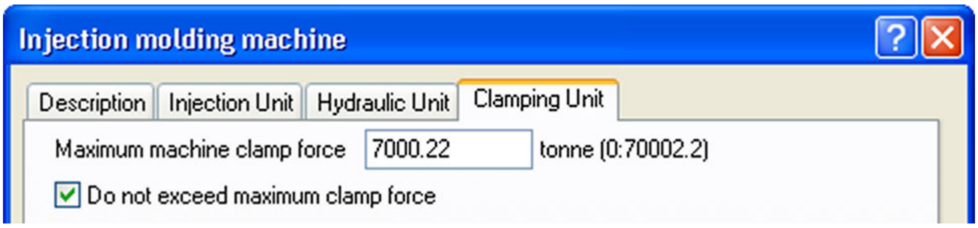


Figure 502: Injection molding machine dialog, hydraulic unit tab

Mold material

The mold material specified here is the global setting. If any entity does not have a mold material specifically defined, the material specified here will be used. The default mold material is P-20 tool steel. The mold material properties are used for the heat transfer calculations in the flow solver. If the mold is made of a material other than P20, the material can be chosen from the mold material database, or the material can be edited. If the mold is made of a grade of steel, P-20 is an acceptable material to use because the conductivities are similar enough so the temperature uniformity is close for all steels.

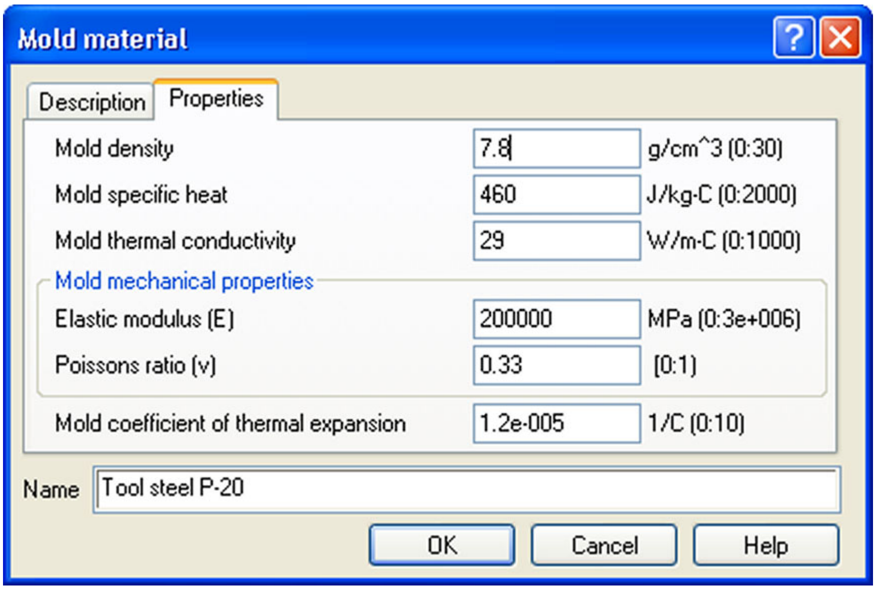


Figure 503: Mold material dialog, P20's properties

Solver Parameters

The parameters available are related to the mesh type. With one exception, Midplane and Fusion are the same. 3D meshes have many different options. Table 88 shows what parameter categories are available for the mesh types.

Table 88: Solver parameter tabs by mesh type

Parameter tab	Midplane	Fusion	3D
Mesh/Boundary	✓	✓	
Intermediate Output	✓	✓	
Convergence	✓	✓	
Restart	✓	✓	
Fiber Analysis	✓	✓	✓
Core Shift	✓	✓	✓
Interface	✓		
Flow Analysis			✓
Cool Analysis			✓
Mesh			✓

Mesh/Boundary (Midplane / Fusion)

Number of laminates across the thickness

The number of laminae across thickness defines the number of division the plastic cross section is broken into. Values that can be specified are 8, 10, 12, 14, 16, 18, and 20.

If the analysis sequence is **Cool + Flow** actual number of layers used in the analysis is the specified value. This is called an **asymmetric** analysis. The mold temperatures are not uniform on the element or matched set of elements. If a fill or flow analysis is done without cooling as an input, one-half of the specified number of layers across the thickness are used. This is called a **symmetric** analysis. Internally defined, non-uniform layer thicknesses are used in the analysis. The interface between the laminae are called grid points. The normalized coordinates of grid points are listed in Table 89 below, where zero is at the center line of the thickness with -1.0 and 1.0 are at the wall.

Variables such as temperature, velocity, shear rate, and viscosity are stored at the grid point of each layer. As the number of layers used in the analysis increases, a more accurate numerical solution is expected. However, the CPU time required to complete the analysis increases significantly as the number of layers increase. The disk space required to store the results will also increase as the number of layers increase.

With triangular elements as shown in Figure 504, the normalize thickness of 0.0 is at the center plane of the thickness. For midplane meshes, 1.0 is at the positive mold wall, and -1.0 is at the negative mold wall. The positive side is in the positive direction of the normal to the element, defined by the element connectivity and the right-hand rule. For Fusion meshes, the normalize thickness of 0.0 is the center line, 1.0 is the element viewed or referred to, and -1.0 is the matched element.

Note that the local x-coordinate is along the direction of node 1 (lowest node number) pointing toward node 2 (middle node number). The y-coordinate lies on the plane defined by nodes 1, 2, and 3, and points toward node 3 perpendicular to the x-coordinate. The z-coordinate (or the element normal) is defined by the cross product of the local x- and y-coordinates.

With axisymmetric one-dimensional elements, as shown in Figure 505 only half of the specified layers are used in the analysis. In this case, the normalize thickness of 0.0 is at the center line of the one-dimensional element, and 1.0 is at the mold wall, as shown below. Note that the local x-coordinate of the one-dimensional element lies along the line defined by nodes 1 and 2, and the direction of the x-coordinate is from node 1 toward node 2.

Table 89: Laminate normalized thickness

Number of layers across the thickness						
8	10	12	14	16	18	20
						1.000
					1.000	0.938
				1.000	0.926	0.864
			1.000	0.914	0.840	0.779
		1.000	0.900	0.816	0.743	0.685
	1.000	0.880	0.784	0.706	0.636	0.583
1.000	0.856	0.738	0.653	0.585	0.520	0.474
0.816	0.681	0.577	0.508	0.453	0.397	0.360
0.586	0.477	0.399	0.350	0.310	0.268	0.243
0.313	0.248	0.206	0.180	0.158	0.135	0.123
0.000	0.000	0.000	0.000	0.000	0.000	0.000

Table 89: Laminate normalized thickness

Number of layers across the thickness						
8	10	12	14	16	18	20
-0.313	-0.248	-0.206	-0.180	-0.158	-0.135	-0.123
-0.586	-0.477	-0.399	-0.350	-0.310	-0.268	-0.243
-0.816	-0.681	-0.577	-0.508	-0.453	-0.397	-0.360
-1.000	-0.856	-0.738	-0.653	-0.585	-0.520	-0.474
	-1.000	-0.880	-0.784	-0.706	-0.636	-0.583
		-1.000	-0.900	-0.816	-0.743	-0.685
			-1.000	-0.914	-0.840	-0.779
				-1.000	-0.926	-0.864
					-1.000	-0.938
						-1.000

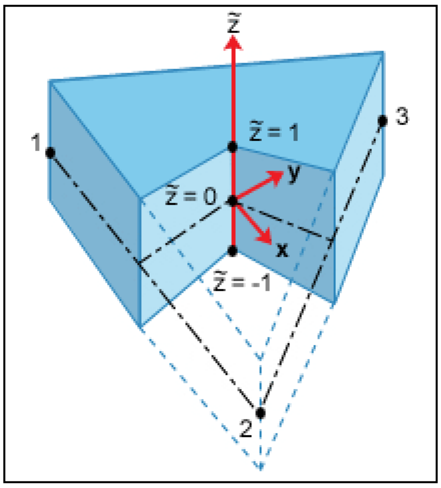


Figure 504: Triangular element definition

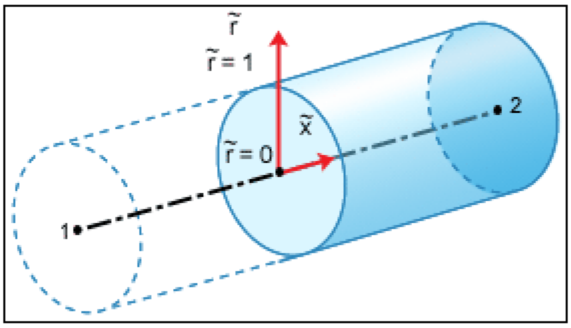


Figure 505: Beam element definition

Mold-melt heat transfer coefficient

The heat transfer coefficient (HTC) defines at the interface between the melt and the mold wall, which models the associated heat resistance. If the HTC is zero, there is no heat exchange between the melt and mold wall. If h nears infinity, then there is perfect thermal contact between the melt and mold wall. The HTC has different values depending on the phase of molding. During the filling phase, the HTC is 5000 W/m^{2}C, then during packing the HTC drops again to 2500 W/m^{2}C. Finally during cooling when the pressure in an element drops to zero (the part detaches from the mold wall,) the HTC drops again to 1250 W/m^{2}C. These values can be changed. A distinction is made between the cavity and core side when the elements are detached. The cavity and core side of an element is defined in the element property. An example for midplane elements is shown in Figure 507. For Fusion, there are only two choices, Top=Cavity and Top=Core. These assignments only need to be done if the HTC is different between the cavity and core. Generally default values are used.

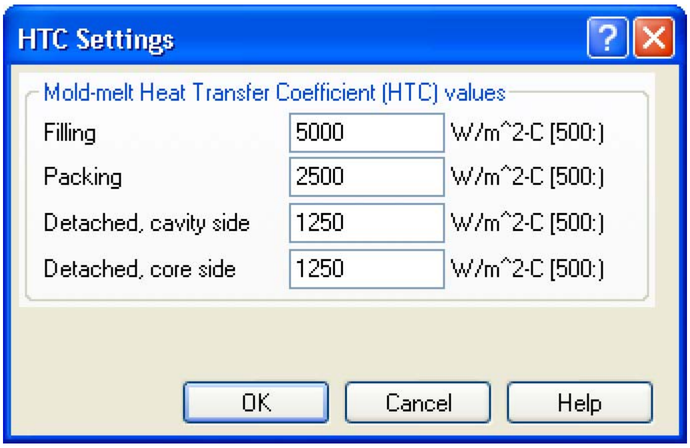


Figure 506: Heat transfer coefficient settings

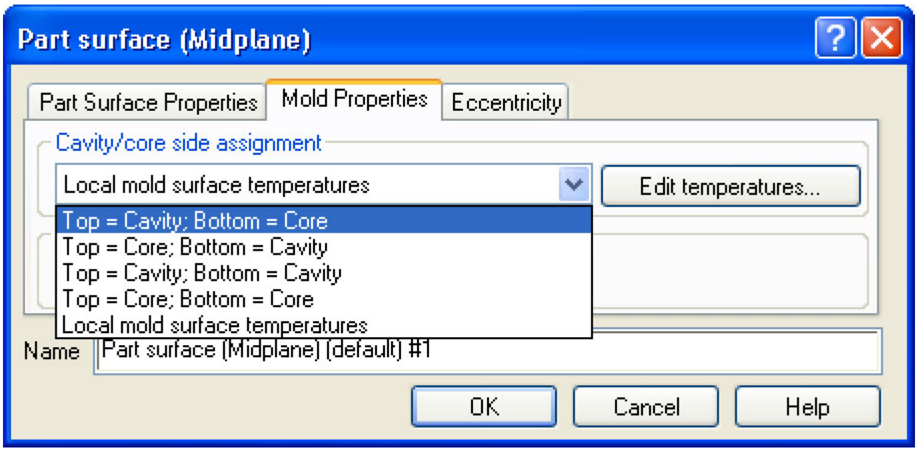


Figure 507: Part surface (midplane) dialog with cavity/core side assignments

Intermediate Output (Midplane / Fusion)

On this tab, the number of intermediate results are available, as shown in Figure 508. There is a checkbox to dynamically update results display during analysis, which is checked by default. As the flow analysis is running, intermediate results are updated so you can see the progress before the analysis is finished.

Intermediate results are broken down into two types and two categories for a total of four different intermediate results. The intermediate results available are:

- Filling phase, Regular result.
- Filling phase, Profiled result.
- Packing phase, Regular result.
- Packing phase, Profiled result.

The regular results record results at various time intervals, and are normally animated through time. Profiled results record information not only through time, but also through the thickness of the plastic cross-section. These results are normally animated through normalized thickness, but can also be animated through time.

Regular or profiled results have 3 options on when the results are written, including:

- None, default setting for profiled results.
- Write at constant intervals, default setting for regular results.
- Write at specified times.

The profiled results default to zero because they take a significant amount of disk space to store all the information. If you are interested in looking at profiled information, you will need to specify the results that you want.

When specifying the number of intervals required, the number you can specify depends on what type of result it is. For regular intermediate results, you can specify the following number of results, 0, 4, 8, 12, 20, 40 and 100. For profiled results you can specify 0, 4, 8, 12, and 20.

When writing specified times, you can list the times at which you want results recorded.

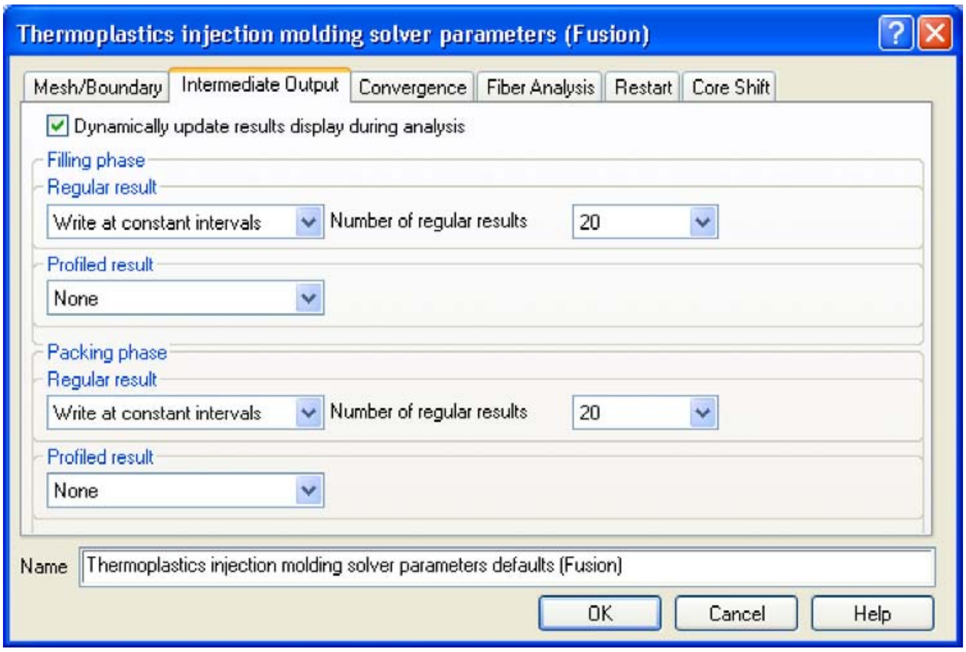


Figure 508: Flow solver parameters (Fusion), intermediate output with default settings

Convergence (Midplane / Fusion)

On the Convergence tab, the settings include:

- Flow rate convergence tolerance.
- Maximum number of flow rate iterations.
- Melt temperature convergence tolerance.
- Maximum number of melt temperature iterations.
- Nodal growth mechanism.
- Viscosity treatment at high shear rates.

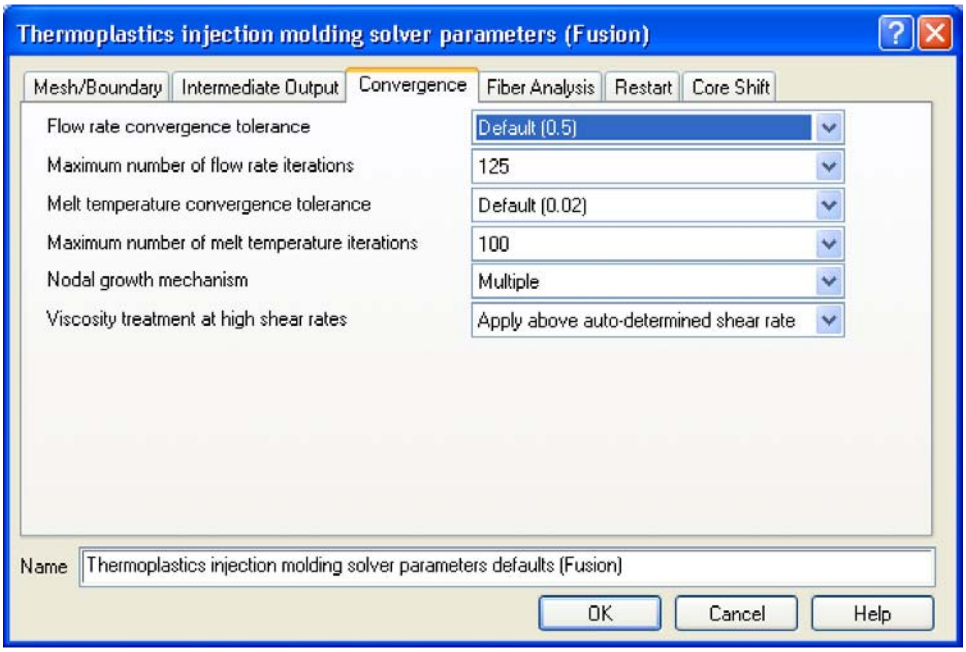


Figure 509: Flow solver parameters (Fusion), Convergence with default settings

Flow rate convergence tolerance

Convergence tolerances apply to the % change in a function value from one iteration to the next, and are used to identify when a solution has converged. As soon as this % change value falls below the convergence tolerance, the solution has converged.

There are 3 available settings:

- Tight (0.05).
- Default (0.5).
- Loose (1.8).

Tightening a convergence tolerance may improve the solution accuracy, but will increase analysis time and may lead to convergence problem warnings. Also, if the analysis is generating convergence problem warnings, try loosening the tolerance to assist the solver in completing the simulation.

Typically, there is no need to change the default value.

Maximum number of flow rate iterations

Specifies the maximum number of iterations that the program will perform in order to solve the flow rate equations. The program will continue until either the maximum number of iterations have been surpassed, or the error limit is less than the specified value. The default value is 125. The number of iterations can be increased to 250.

Typically, there is no need to change the default value.

Melt temperature convergence tolerance

The melt temperature convergence tolerance applies to the % change in the temperature calculations from one iteration to the next, and is used to identify when a solution has converged. As soon as this % change value falls below the convergence tolerance, the solution has converged.

There are 3 available settings:

- Tight (0.02).
- Default (0.2).
- Loose (1.8).

Tightening a convergence tolerance may improve the solution accuracy, but will increase analysis time and may lead to convergence problem warnings. Also, if the analysis is generating convergence problem warnings, try loosening the tolerance to assist the solver in completing the simulation.

Typically, there is no need to change the default value.

Maximum number of melt temperature iterations

This parameter specifies the maximum number of iterations that the program will perform in order to solve the melt temperature equations. There are two choices, 100 (default), and 200. The program will continue until either the maximum number of iterations has been surpassed, or the error limit is less than the specified value.

Typically, there is no need to change the default value.

Nodal growth mechanism

The nodal growth mechanism determines how many nodes are added to the flow front calculations. The default is **Multiple**. With this choice, more than one node can be added to the calculations at any growing step. The number added is determined by the amount of time required to fill the nodes to be added. Nodal growth can be set to Single. At any time step, only one node will be added. This can be make the results more accurate, but at the price of compute time.

Typically, there is no need to change the default value.

Viscosity Treatment at High Shear Rates

For certain materials with high temperature or shear sensitivity, extrapolation of viscosity values to high shear rates may introduce instability in the flow calculations. For example, this instability may result in the prediction of an asymmetric filling pattern where symmetric filling is expected.

This option relates to a corrective algorithm that eliminates the above-mentioned instability at high shear rates. The algorithm significantly reduces the rate at which viscosity falls at very high shear rates. The threshold of the correction is automatically determined.

The following settings are available:

- Do not apply.
  - No high shear rate viscosity treatment will be applied. Select this option if you know that the materials you analyze do not exhibit high temperature or shear sensitivity.
- Apply it above auto-determined shear rate.
  - The solver will apply the corrective algorithm if high shear rates are predicted in the analysis.

Restart (Midplane / Fusion)

On the Restart page, you can specify the number of restart files that are written. A restart file is used to restart an analysis if it was accidentally stopped. The defaults are 0. Generally restart files are not created.

Fiber Analysis (Midplane / Fusion / 3D)

The settings on this tab are used to change coefficients used within various fiber orientation models, and to change the models themselves. The default values should be used. These options are here primarily for research with Fiber orientation. The help files in Synergy have a significant amount of detail related to this information.

Core Shift (Midplane / Fusion / 3D)

A flow analysis can be run simultaneously with a warp analysis to determine the deflection on a core. Both a flow and warp licenses are required. The core must be made of tetrahedral elements and constrained. Figure 510 shows the parameters for a Fusion core shift analysis. Midplane meshes have the same parameters as Fusion. A 3D mesh has a different layout for the core shift tab. The 3D core shift tab is shown in Figure 511.

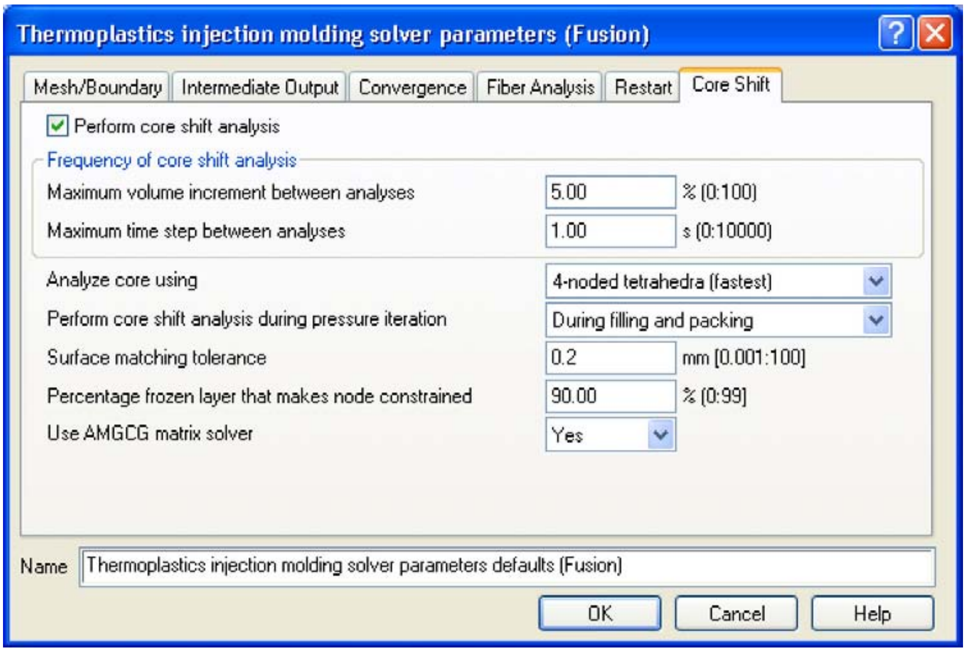


Figure 510: Flow solver parameters (Fusion), Core shift with default settings

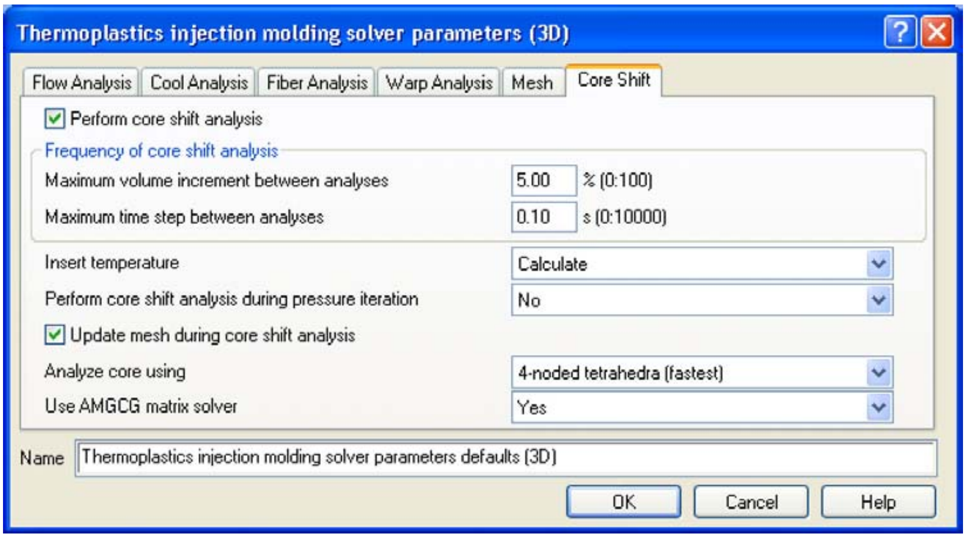


Figure 511: Flow solver parameters (3D), Core shift with default settings

Perform core shift analysis (Midplane / Fusion / 3D)

To run the analysis, the box **Perform core shift analysis** must be checked, and is checked by default.

Frequency of core shift analysis (Midplane / Fusion / 3D)

By default, the frequency is set to 5% of the part volume, and 1.0 seconds. The time increment will work for both the filling and packing phases. The default values are reasonable for preliminary work, but the values may need to be smaller to be most accurate. One indication that the increments are to small is if the core wobbles.

Analyze core using (Midplane / Fusion / 3D)

This setting determines the type of element to be used. The default is a 4-noded tetrahedral element. This will give reasonable answers if the core does not deflect much. The other choice is a 10-noded tetrahedral element. A 10-noded element should be used if the deflection is large and for more accurate results.

Perform core shift analysis during pressure iteration (Midplane / Fusion / 3D)

During the pressure iterations of the flow analysis, the pressure will change a bit. This influences the core deflection analysis. There are three options including:

- No.
  - This will limit the number of core deflection iterations to the number specified in the frequency settings.
- During packing only.
  - This will add core shift calculations for the packing phase only.
- During packing and filling.
  - This will add core shift calculations for the filling packing phases.

The last two options will produce the most accurate results, but will also significantly increase the compute time.

Surface matching tolerance (Midplane / Fusion)

It is difficult to get a perfect match between the plastic elements and the tetrahedral core elements. The core and plastic elements are considered touching if the distance between the core and plastic are less than the tolerance. The default value is 0.2 mm.

Percentage frozen layer that makes node constrained (Midplane / Fusion)

This determines the percentage of the cross section that must be frozen in order to make the core ridged. The default value is 90%. This may be a bit high. If the deflections are greater than you expect, lower the value to see the influence on the core.

Use AMGCG matrix solver (Midplane / Fusion / 3D)

The AMGCG matrix solver makes the deflection calculations go much faster without sacrificing accuracy. The default is on, and should be left on.

Insert temperature (3D)

This option specifies whether the insert temperatures are set to a fixed value, or calculated by the flow analysis. The following settings are available:

- Calculate.
  - Select this option if you do not want the simulation to assume that the insert temperature remains constant throughout the molding cycle. This setting increases the analysis time, but will only provide more accurate results if the quality of the insert mesh is good.
- Fix at mold temperature for core-shift analysis.
  - Select this option if you want the simulation to set the insert temperature to a constant equal to the mold temperature specified in the process settings, but only when performing a core-shift analysis.
- Fix at mold temperature for any analysis.
  - Select this option if you want the simulation to set the insert temperature to a constant equal to the mold temperature specified in the process settings, for all flow analyses.

Update mesh during core shift analysis (3D)

When the predicted core shift is large, the resulting mesh distortion may make the flow analysis unstable, and the analysis will stop. If this is the case, turning this option off will allow the analysis to complete, but the solution will be somewhat less accurate.

Interface (Midplane)

This tab enables you to set the parameters for interface files to ABAQUS or LS-DYNA. Flow results are written to these interface files so a warpage and stress analysis can be done in these CAE codes.

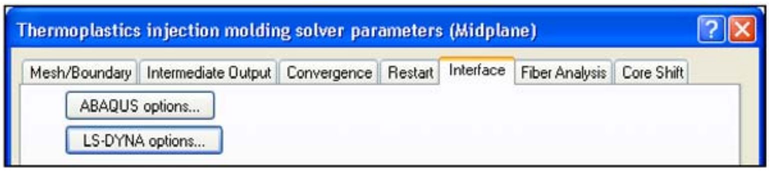


Figure 512: Flow solver parameters (Midplane), Interface tab

Flow Analysis (3D)

The Flow analysis tab, shown in Figure 513, has three parameters to set:

- Solver setup.
- Intermediate results
- Recovery data.

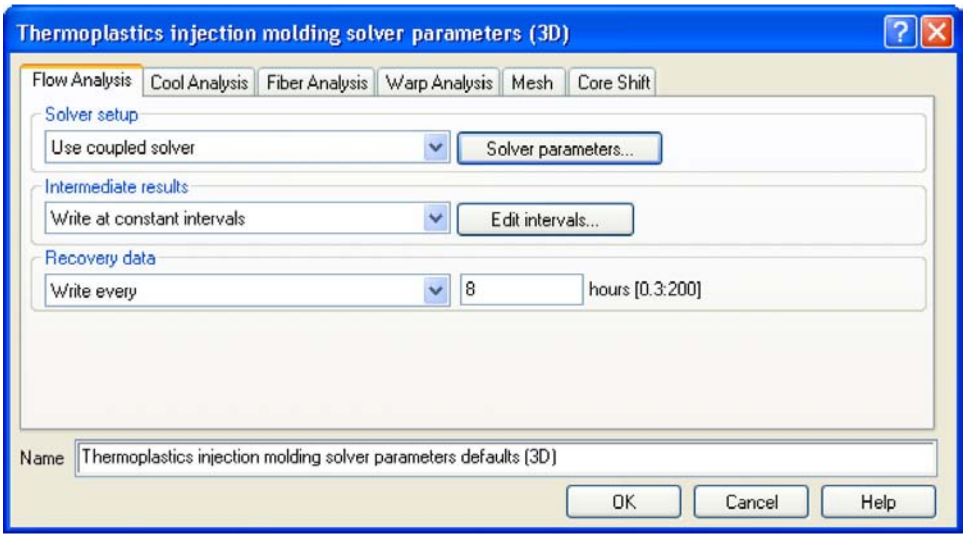


Figure 513: Flow solver parameters (3D), flow analysis tab

Solver setup

There are two solver choices, including:

- Use coupled solver.
  - The coupled solver is the default. It is called coupled because the pressure and velocity are combined into one matrix and are solved simultaneously. This is the most accurate and fastest solver, and is the default.
  - By clicking the Solver parameters, boxes are available for simulating inertia and gravity. They are unchecked by default. Inertia is important if there are significant and rapid changes in wall thickness. With inertia on, a fine mesh and many intermediate results, jetting can be predicted.
  - The coupled solver uses Navier-Stokes equations. The classical complete Navier-Stokes equations (including inertia) for non-Newtonian viscosity is the slowest but most complete option. Inertia is the acceleration term in the momentum equations. This means that if fluid has a velocity, it will tend to keep that velocity, unless some other force acts upon it. Other forces can include the viscous forces

(stresses) which come from shear deformation. In the particular case of injection molding of polymers, the viscous stresses are very large compared to the inertia terms. This is because of the relatively high viscosity of polymer melt (compared to other fluids such as air) and the narrow cavities through which polymers are injected. This is equivalent to saying: In injection molding the Reynold's number of the flow is usually much less than one and so inertia terms are not significant. Generally speaking, the inertia option should be used when the Reynold's number is expected to be greater than 1. Even then, you should consider whether your analysis needs to have this accuracy. There might be high velocities in a small gate region, but if the gate is only a small contribution to the total injection pressure, there may be little difference in results with or without inertia.

- Navier-Stokes without inertia.
- If the inertia terms were removed from the momentum equations, the calculations are simplified a little, resulting in a faster analysis. Since in most cases, the dropping of the inertia terms will make no difference to injection molding predictions, this option is a good choice for most users. A Navier-Stokes analysis without inertia terms is sometimes called a Stokes analysis. The speed saving is about 10-30%.

- Use segregated solver.
  - This is the original 3D solver that has been replaced by the coupled solver.

To illustrate the difference between the 3D coupled solver, with and without inertia, look at Figure 514. Both parts are using the result **Polymer fill region**. The top plot shows the filling of the part without taking into account inertia. The bottom plot considers inertia. The bottom part fills from the end of the part to the gate. The analysis time for the bottom plot was nearly 8 times that of the top plot. To achive the jetting prediction, there was 25 intermediate steps and there was a very fine mesh.

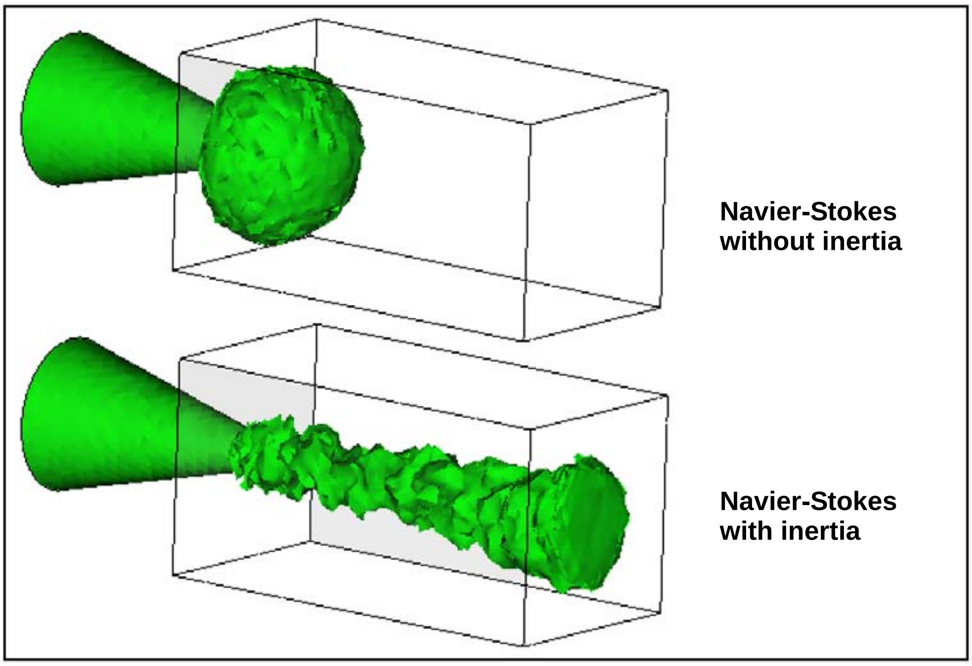


Figure 514: Filling pattern with and without inertia on a 3D part

Intermediate results

By default, MPI/Flow3D will write 5 intermediate results at constant time intervals during the filling and packing phases of the analysis. These result sets serve a number of purposes:

3. They allow you to view most result types, for example, temperature, pressure, etc. before the part is fully filled or fully packed, and so provides an opportunity to abort the analysis early if the filling or packing is not proceeding as desired.
4. They provide the "time slices" for results that you can animate through filling or packing time.
5. They provide the data for XY plots of a result vs. time, e.g. pressure vs. time, for selected nodes in the model.
6. They enable the restart capability as described below.

The default value of 5 intermediate results is often not enough to catch the intermediate information desired. Consider using more steps for the filling. In Figure 515, there are many filling phase results in order to capture an jetting that may occur. For the packing, determine the number of intermediate steps by the total packing + cooling time. If this time is 25 seconds, request 25 intermediate steps to get results every second.

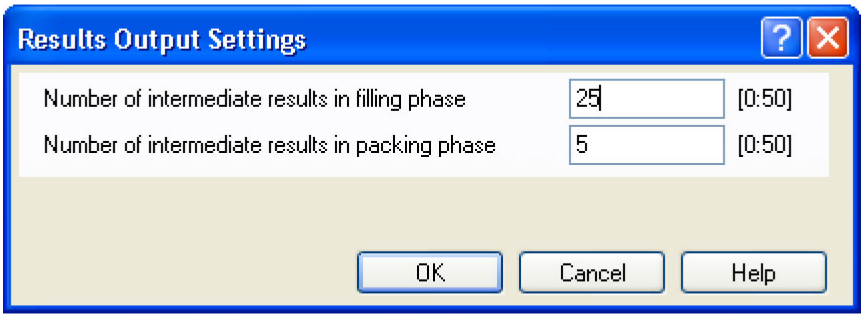


Figure 515: 3D Results output settings

Recovery data

By default, MPI/Flow3D will write recovery data after every 8 hours of analysis. Recovery data is used to safeguard against system failure by providing a full-detail snapshot of the simulation data in memory at the time of writing. In the event that system failure does occur and recovery data is available, the MPI/Flow3D solver will be able to quickly restart the analysis from the point at which the last set of recovery data was written. A 3D flow analysis now will generally take less than 8 hours to run. If you think you may need to restart the analysis, set this to an appropriate time.

Cool Analysis (3D)

The cool analysis parameters are discussed in the advanced cooling class. The set how the cooling analysis will run. The defaults are normally used. Possibly, the box **Calculate internal mold temperatures** is checked. This will create a result so a cutting plane can be move through the mold to see temperature gradients within the mold.

Mesh (3D)

The Mesh tab, shown in Figure 516, has three sets of parameters including:

- Finite difference grid parameters for beam elements.
    - The number of laminates in the polymer and mold are indicated plus the thickness ratio between a plastic laminate and mold laminate. The default values work well.
- Overmolding interface tolerance.
    - This option relates to the method by which the solver identifies which nodes lie at the interface between the cavity and insert/overmolded part. Where flat surfaces meet at the interface, the geometrical situation is not problematic. However, when curved surfaces meet, the faceted nature of the mesh on the two surfaces will result in gaps, or possibly even overlaps, and so make it more difficult to determine which nodes are at the interface.
    - The interface tolerance value determines how far a node is allowed to be located from the expected interface position and still be regarded as at the interface. The tolerance settings are derived by examining the edge length values all of the surface facets on the tetrahedral mesh of both parts to determine a minimum, average and maximum distance value.
    - The default setting will be suitable for most models. For models with highly curved and coarsely faceted surfaces, it may be necessary to increase the interface tolerance value from the minimum to the average or maximum facet length setting.
- Overmolding interface temperature solution method.
    - The methods include, Constraint transformation (default) and flux transformation.

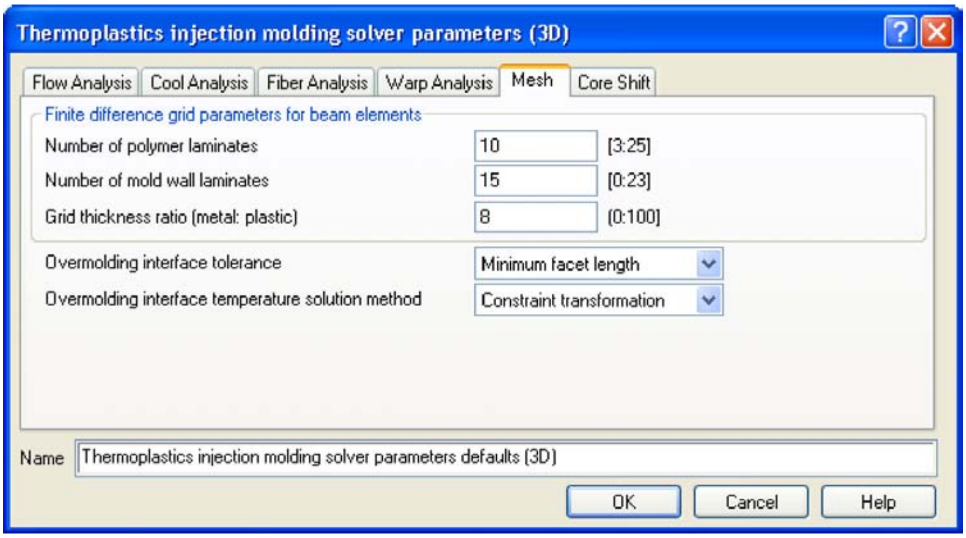


Figure 516: Flow solver parameters (3D), mesh tab

Differences between 3D and Fusion/midplane analysis

Midplane and Fusion analyses

Fusion and Midplane flow analysis, so-called 2.5D analysis, make the following assumptions when solving the mass, momentum and energy equations used to simulate the injection molding process:

- The Hele-Shaw assumption is applicable, i.e.
  - No pressure variation in thickness direction.
  - Velocity can be derived from pressure gradient so only pressure and temperature need to be calculated at each node.
  - Laminar flow.
- In-plane conduction can be ignored.
- Heat loss from edges can be ignored.
- Inertia and gravity effects can be ignored.

3D analysis

3D analysis on the other hand makes fewer simplifying assumptions and:

- Solves for pressure, temperature and the 3 velocity components at each node.
- Considers heat conduction in all directions.
- Provides option to consider inertia and/or gravity effects.

When to use 3D analysis

The following are some of the possible situations where 3D analysis may be preferable over a 2.5D analysis:

- Filling and packing of thick-walled parts.
- Parts with complex geometry, that is, where a midplane representation is not possible or where there are sudden changes in thickness.
- Where 3D cooling effects are anticipated in the gate region or other parts of the mold.
- Inertia or gravity effects are anticipated to be non-negligible.
- Where flow imbalance is observed in symmetric multi-cavity feed systems.

Injection location assignment

With a 3D mesh, the area of the injection location is the total area of the elements the injection locations touch as shown in Figure 517. On parts with fine meshes, several injection locations may need to be set to ensure a reasonable area. If the part has a 3D runner system, all the internal nodes at the end of the feed system must be selected, as shown in Figure 518.

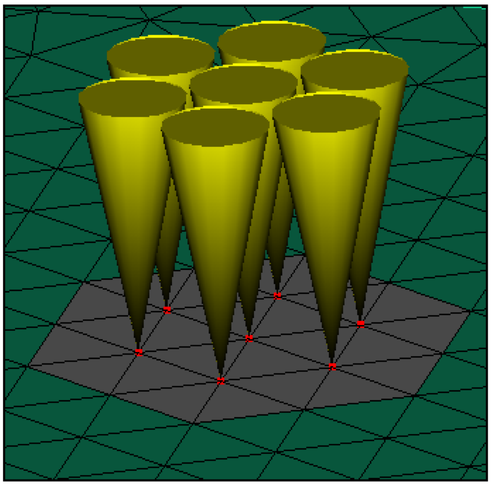


Figure 517: Gates on a 3D part

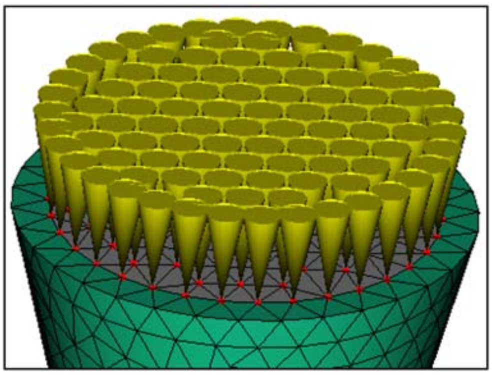


Figure 518: Gates at the end of the feed system

What You've Learned

There are many parameters that can be set when running a flow analysis. Most have default values that can be used. If you need to change the default settings care should be taken to understand what affect the change will have on the analysis.

The primary process settings wizard's Flow settings page gives you access to all the parameters that you can set. The main parameters include:

- Mold surface temperature.
- Melt temperature.
- Filling control.
- Velocity/pressure switch-over.
- Pack/holding control.
- Cooling time.

The Advanced options in many cases don't need to be changed. It is within the advanced options the following are set:

- Molding material properties.
- Process controller.
- Injection molding machine.
- Mold Material.
- Solver Parameters.

  Mesh/Boundary (Midplane / Fusion).

  Intermediate output (Midplane / Fusion).

  Convergence (Midplane / Fusion).

  Restart (Midplane / Fusion).

  Fiber Analysis (Midplane / Fusion /3D).

  Core shift (Midplane / Fusion /3D).

  Interface (Midplane).

  Flow Analysis (3D).

  Cool analysis (3D).

  Mesh (3D).

CHAPTER 22

Creating Reports

Aim

The aim of this chapter is to learn about the features of the Report Generation Wizard to quickly make HTML, Power Point or Word Document reports.

Why do it

Using the Report Generation wizard is a very quick and easy way to create reports that can then be sent to anyone with an Internet browser or Microsoft office (Word or Powerpoint). There are a few options that can make the report easier to generate and more customizable which you will learn after completing this chapter.

Overview

The basic process for creating a report include the following steps:

1. Start the Report Generation Wizard.
2. Pick studies to include.
3. Pick the plots to include.
4. Select the format for the report.
5. Generate the report.

This chapter will take these simple steps and expand them.

Theory and Concepts - Chapter title

Starting the Report Generation Wizard

The Report Generation Wizard quickly creates results reports. The wizard is used to create a report including any and all studies in the project directory the wizard is started in. The wizard is accessed in the **Report ➨ Report Generation Wizard** menu or in the Project Pane context menu.

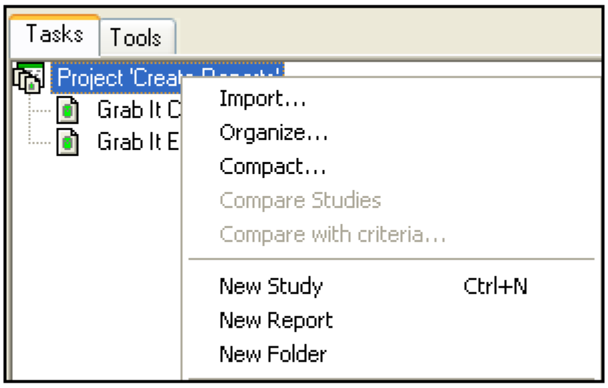


Figure 519: The Project Pane Context Menu

Selecting Studies

All studies in the project directory are listed in the Study Selection page of the wizard shown in Figure 520. Any studies that are currently open when you start the wizard are automatically selected. Use the **Add** and **Remove** buttons as necessary to move studies from one field to another.

MPI will list allow you to create reports of studies inside the same project.

Add the study(ies) to the same project to be able to create a report from studies originally not inside the same project.

To select more than one study at a time, use Windows standards. The **Ctrl** key is depressed to select one study at a time, hold down the **Shift** key to select a range.

Generally it is best to have the studies open before the wizard is started. This makes it easier to ensure the correct plot setting, rotations and magnifications are correct before the report is created.

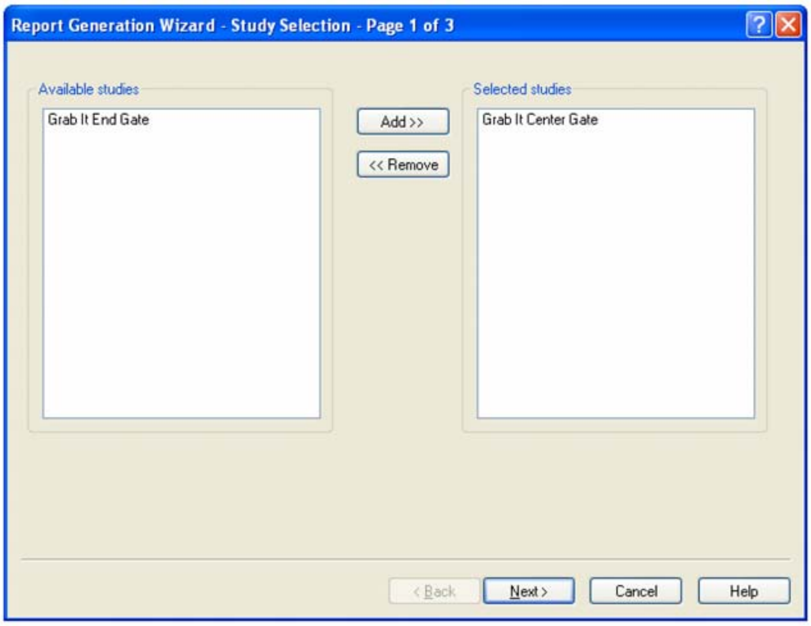


Figure 520: The study selection dialog

Results included in the report

Once you have chosen which studies are to be included in the report, you can choose the individual plots within the studies. The Study drop-down list allows you to pick the study you want to pick plots for.

For each study, the available plots are listed. This includes text file results such as the Results summary file. Highlight the plot or plots to be used and click the **Add** button. The order of the plots does not matter because you can change this later.

Text results

Most of the items in the list are graphic results. However, the text files based results are also available. Text files listed on the navigation pane (Table of Contents) display to the user a link. When a link on the table of contents is clicked, the report display area changes to only display the contents of the text file. To get back to the main part of the report, click any link on the table of contents.

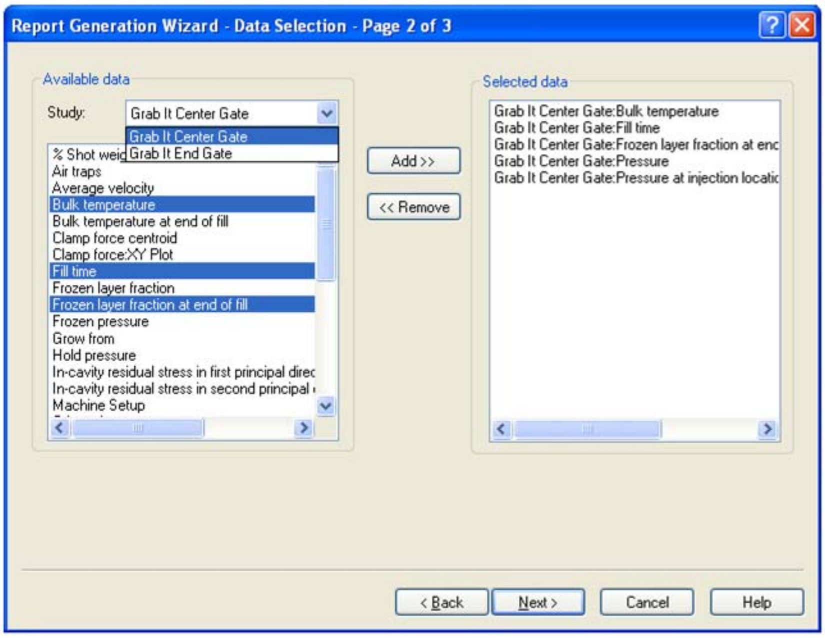


Figure 521: The plot data selection dialog

Format the report

The last page of the wizard is the **Report Layout** page. This is where each plot and the overall look of the report are formatted. There are several ways formatting can take place.

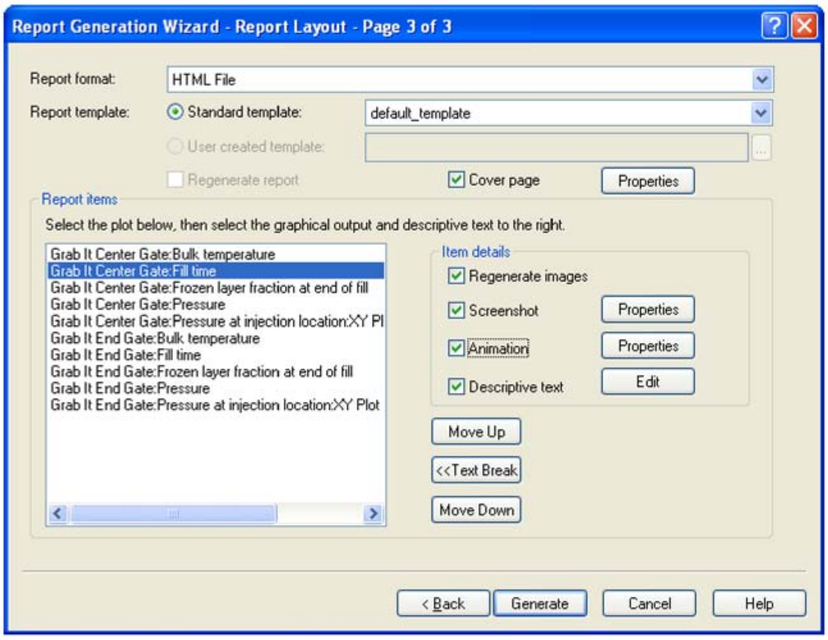


Figure 522: The report layout dialog

Choosing the report format

MPI can produce reports in three formats:

- HTML.
- Microsoft Word.
- Microsoft PowerPoint.

Select the format on the third page of the Report Generation wizard, as shown in Figure 523. You can build a report in HTML initially for convenience, then change the format once you are happy with the images.

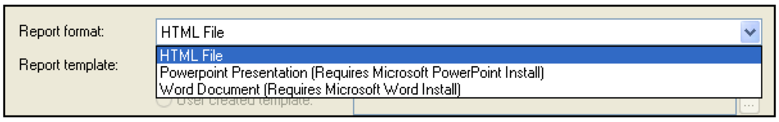


Figure 523: Report generator showing report formats available

Output template

The output template contains the background of the report. There are three choices:

- Contemporary.
- Default.
- Notebook.

You can change which template is selected at any time before you generate a report. If you want to change the template after you have generated a report, you have to format the report again.

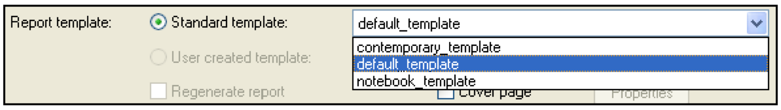


Figure 524: Report generator showing report templates available

Cover page

The cover page contains information such as:

- Project title.
- Prepared by.
- Requested by.
- Reviewed by.
- Company logo.
- Cover picture.

The information can be entered as necessary and the cover page can be turned on or off as needed from the Report Layout page.

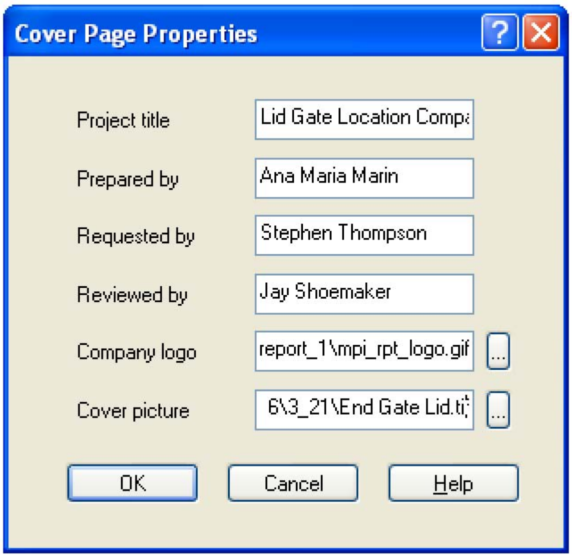


Figure 525: Cover page properties dialog

Plot formatting

You can adjust the appearance of every plot in the report. The report generator uses the plot properties set in Synergy. For instance, if you had contour lines set for the method of display this would be used in the report. Every plot that you include should have the plot properties set. You can use the Report Generation wizard to specify a static screen shot or an animation and if you want descriptive text or not. See Figure 526.

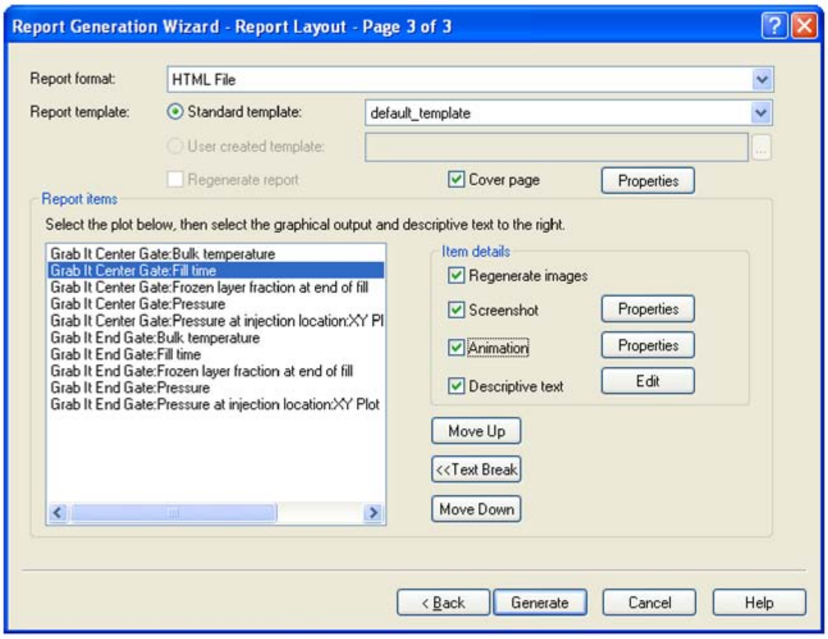


Figure 526: Plot formatting dialog

Screen shot properties

For screen shots and animations the size and rotation of the image can be set as shown in Figure 527.

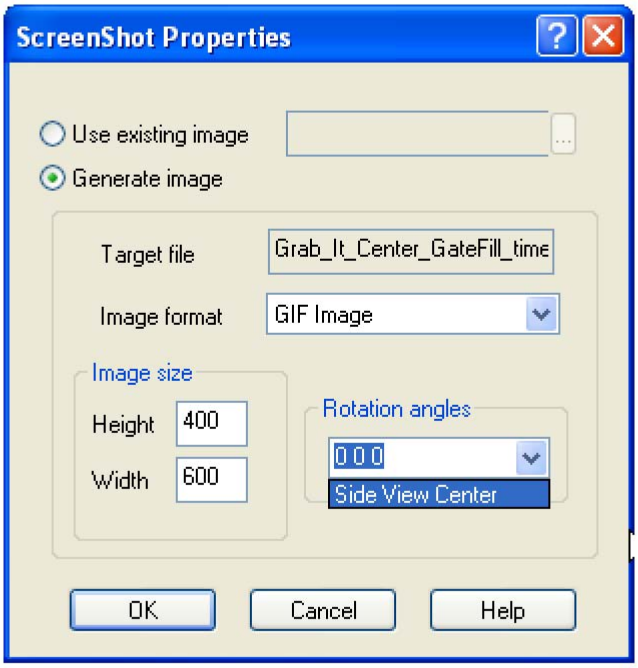


Figure 527: Screen Shot Properties dialog

Use existing image

Rather than having Synergy control the image, you can use an existing image.

1. Click on the **Use existing image** button
2. Browse to find the image you want.

This is a popular option, because it gives you the most flexibility in the creation of the report. Plots are set up exactly as you want them to be then the image can be saved. The image will be whatever size the window is. Normally, the graphics window containing the image you want will need to be made smaller so the image will be an appropriate size in the report. To save an image from SYNERGY, use **Edit ➨ Save image to file** or **Ctrl + F.**

Saving the file in GIF or JPEG format is preferred over bitmaps because bitmaps take a lot of file space.

If you create your own images, the names of the plots as shown in Figure 527, are just links from the navigation pane in the report to the image you import. The name of the link can be changed by double-clicking on the link name then entering in the new name for the link.

Adding text

All plots have the option of adding text to the report. To add text, click the **Descriptive text** check box to open the dialog. Text can be added directly into the box or can be pasted in from another source.

When this dialog is open, you cannot manipulate the model. As a result it is sometimes difficult to enter all the text in one session, so using a text editor is useful.

An easy way to add text to a report is to have a text editor open. Size SYNERGY and the text editor on the screen so you can see both at the same time. This way you can add text as you go along and when you are ready to write the report you can paste the text in.

You may also add Plot notes (Right click over a result from the results list) and enter your comments while you are reviewing the results. What you entered as Plot notes will be included in the report as the description text section.

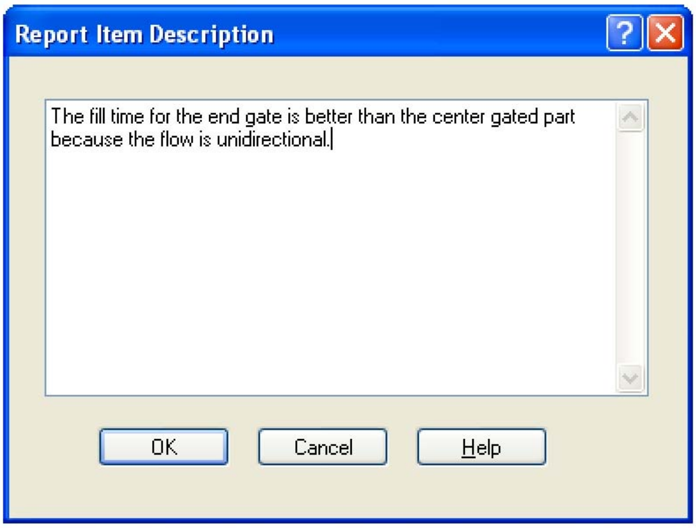


Figure 528: Descriptive text dialog

Report generation

Generate the report with the **Generate** button on the Report Layout page of the wizard. When the report has been generated, it appears in a Synergy window. Browse through the report as if it were a Web site. To look at the report at a later time, use the click **Report** ➡ **View**. The report can be inspected to see if any changes need to be made.

You can generate the report at any time, to check how it will look.

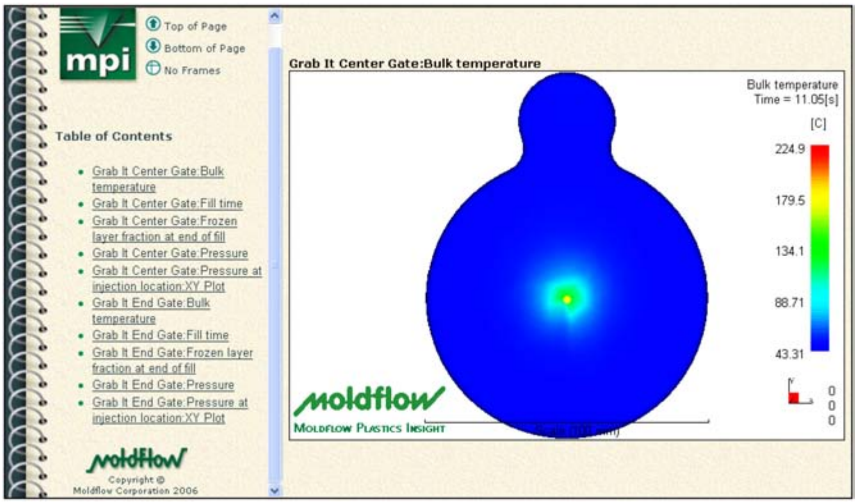


Figure 529: MPI Report (HTML format)

The report generated in MPI (HTML format) is not intended to be printed. If you would like to print the report select the Word or Powerpoint format option instead.

Edit the report

Once a report has been generated for the first time, you can add more components to the report. This can be done through the **Report** menu.

As is shown in Figure 530, the main components of the report can be added without going into the wizard. This includes:

- A cover page.
- An image.
- An animation.
- A Text.

If items are added via this method, the report will need to be re-generated from the wizard.

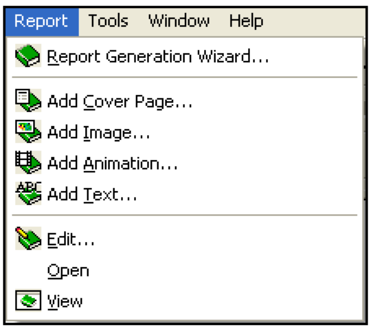


Figure 530: Report Menu dialog

Alternatively, you can create the whole report again via the wizard.

Other generation options

Multiple images and animations can be saved for a result, as shown in Figure 531.

Adding an image or animation

When reviewing a result, set the model as needed for the report. Right click on the result and select **Add Report Image** or **Animation**. If no report exists, Synergy will automatically create an HTML report. Otherwise it will add it to the existing report. If there is more than one report in the project, you are prompted to select the report to add the image to.

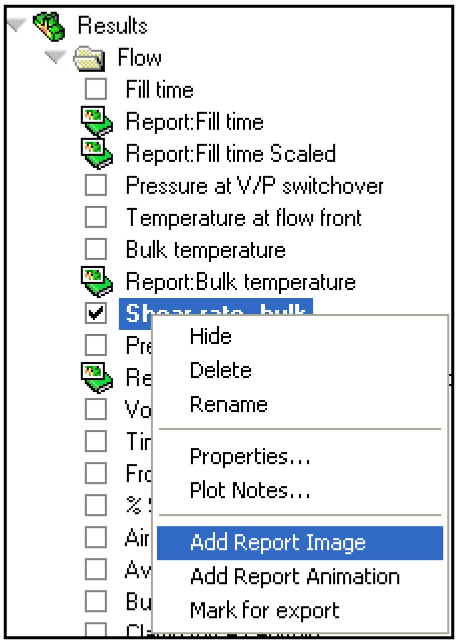


Figure 531: Adding an image for a result

Modifying an image

Once an image is created, the image can be reviewed at any time. Right click on the image to get the context menu shown in Figure 532. From there you can manipulate the image as described in Table 90.

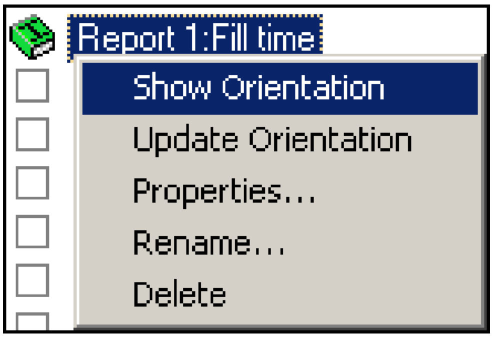


Figure 532: Modifying a report image

Table 90: Report image modification commands

Command	Description
Show Orientation	Rotates the model on the screen to the same rotation as the saved image. The screen magnification is unchanged.
Update Orientation	Updates the image to the rotation and magnification of the image to that of the screen. This allows you to change the existing image.
Properties	Opens the dialog shown in Figure 533. It shows the current settings used for the image. The settings are not changeable. The settings are changed by the update orientation command.
Rename	Opens a dialog where you can set the name and link of the image.
Delete	Deletes the image.

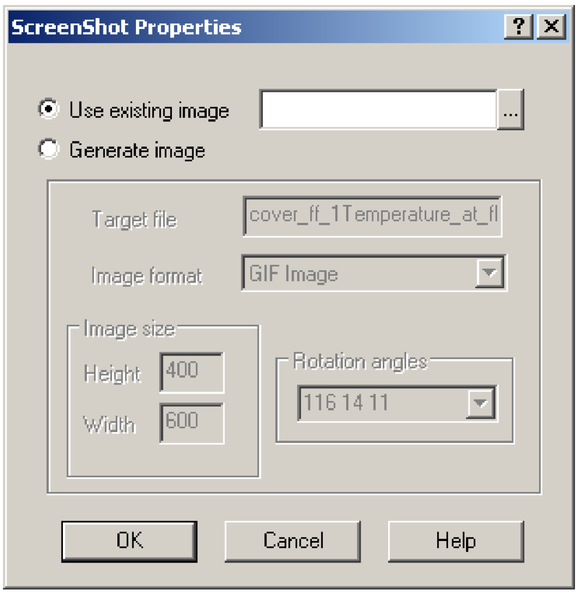


Figure 533: Screenshot Properties dialog for report image modification

Sending the report over the internet

When a report is created and finished, it can be sent to anyone via the Internet or any other method. However, there are many files within a directory structure. To send the report, the files need to be zipped up maintaining the directory structure that is created when the report is first written.

Within Synergy, the report is listed in the **Project View.** The default name is Report, but can be changed.

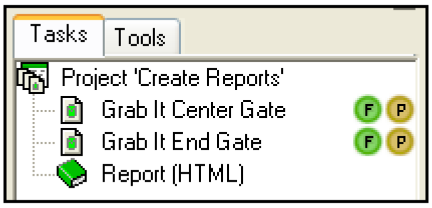


Figure 534: Report List in Synergy

The report on the hard disk has a name of **report**. For additional reports the name is **report~1**, etc. Within the report folder there are a number of standard files and one or more sub folders. All this directory structure must be maintained.

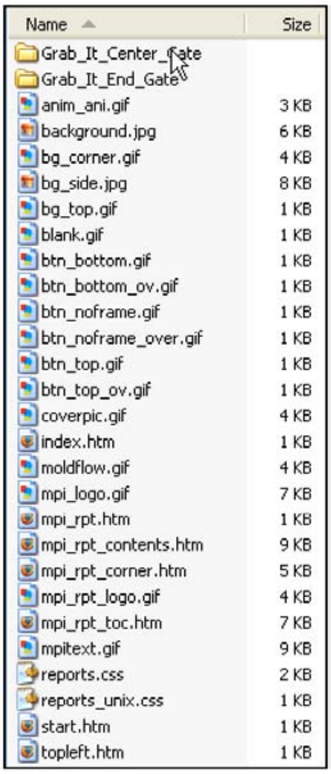


Figure 535: List of files in the report folder (HTML format)

When creating Word or PowerPoints reports they will be saved inside the report folder as well.

When using Winzip make sure that the directory structure is maintained. There are options for zipping the path and sub folders. Both will need to be done. When the zip file of the report is finished, open the zip file to ensure all the files with the path are stored in the zip file.

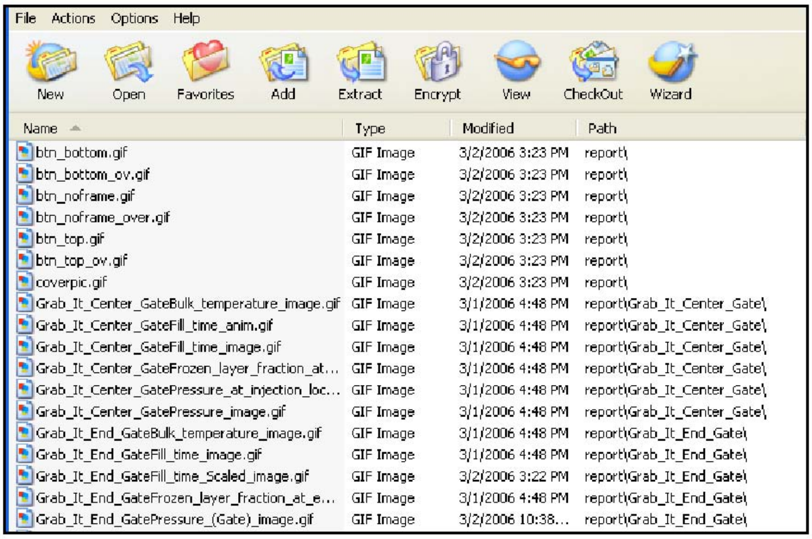


Figure 536: Zip file of a report (HTML format)

Using a report sent over the web

When you receive a report, unzip the report maintaining the directory structure. To open the report in a browser, double click on the file **start.htm**.

What You've Learned

The Report Generation Wizard is an MPI tool used to quickly create reports of one or more study file in the same project folder.

In order to accommodate user preferences, MPI offers the option to create reports in three formats:

- HTML.
- Microsoft Word.
- Microsoft PowerPoint.

The basic steps to follow in order to create a report include:

1. Start the Report Generation Wizard.
2. Pick studies to include.
3. Pick the plots to include.
4. Select the format for the report.
5. Generate the report.

You can modify the report at any time.

CHAPTER 23

Moldflow Communicator

Aim

To learn about the Moldflow Communicator functionality.

Why do it

By understanding the extended functionality that Moldflow Communicator offers, MPI users will be able to decide if using Moldflow Communicator will be more appropriate than an MPI report, for distributing information about a project.

Overview

In this chapter you will learn about the different features of the Moldflow Communicator and you will practice creating and using a Moldflow Communicator results' file. The sections covered in this chapter are:

- Moldflow Communicator introduction.
- Moldflow Communicator interface.
- Using Moldflow Communicator.
- Preparing the Moldflow results file and criteria file in Synergy.

Theory and Concepts - Moldflow Communicator

Introduction

Moldflow Communicator™1.0 is a new, free software product that allows distributed product teams to visualize, quantify and compare Moldflow analysis results. Unlike static 3D viewers, Moldflow Communicator allows users to understand the assumptions behind a given set of analysis results, which is vital to gaining the insight required to make critical design decisions.

One of the greatest challenges faced by CAE companies is ensuring their users properly communicate analysis results and the associated assumptions to members of ever more distributed product development teams. All too often, painstaking design optimization work done at the beginning of the design-to-manufacturing process is not taken advantage of due to poor communication between team members.

Moldflow Communicator allows Moldflow users to easily share the insight gained from analysis-driven design optimization to any member of a product development team, many of whom are not dedicated Moldflow users. Extended team members can now view results dynamically and interactively on 3D models and use a set of powerful tools to compare analysis results "side-by-side" from two iterations to better understand design improvements. One of Moldflow Communicator's most compelling benefits is the ability to understand the assumptions behind a given set of analysis results, which can help team members make informed decisions to reduce development times, improve part quality and speed time to market.

Moldflow Communicator allows companies that either out source or rely on their vendors or suppliers to perform analysis to have a means to quantify the quality of analyses performed. This is achieved through user-specified analysis quality criteria.

Moldflow Communicator Capability

With Moldflow Communicator you can:

- Visualize results.
  - This option allows you to review analysis results on a single model, in more depth compared to the information provided in a static report since you can manipulate the model on your screen by rotating and zooming in.
- Compare results.
  - This option allows you to compare results from two analysis models at the same time. It displays the first result from each study, one above the other while the rotation, zoom and even animations are locked to make the review of results easier.
- Quantify results.
  - This option creates a tabular summary of the studies compared to a set of rules and values (criteria) entered in a criteria file. Wherever the values are different from the criteria or between the studies, the associated table cells are highlighted.

Reduced file sizes

Moldflow Communicator uses a new file format called **Moldflow results file**. This format contains all the model and results information necessary for displaying one or two studies and up to 8 results per study.

User-defined quality criteria

Using a new tool in MPI called the Criteria editor, quality criteria can be set, and used in MPI and Moldflow Communicator to compare studies to the defined criteria.

Compare designs

Studies can be compared to each other and to quality criteria defined by a CAE analyst in MPI. Differences between studies and the quality criteria can easily be found.

Moldflow Communicator Compatibility

Moldflow Communicator can be run by users who do not have a Moldflow product license. Currently, Moldflow Communicator is compatible with results produced by MPI 6.0 Revision 1 or higher.

Files used with Moldflow Communicator

There are two kind of files you can read inside Moldflow Communicator the results file and the criteria file.

- Results file.
    - Moldflow Communicator can view Moldflow Results files (*.mfr) that were created using MPI 6.0 Revision 1 or higher. These Moldflow Results files may contain up to two studies and each of the studies may contain a maximum of eight analysis results.
- Criteria file.
    - A criteria file (*.criteria) can be created and edited in MPI by the CAE analyst and should describe the optimum results and tolerances.

Moldflow Communicator interface

Moldflow Communicator is a simplified version of Synergy (user interface of Moldflow Plastics Insight). Commands can be performed by using the pull down menus, icons or keyboard shortcuts.

The different components of the Moldflow Communicator are displayed in Figure 538.

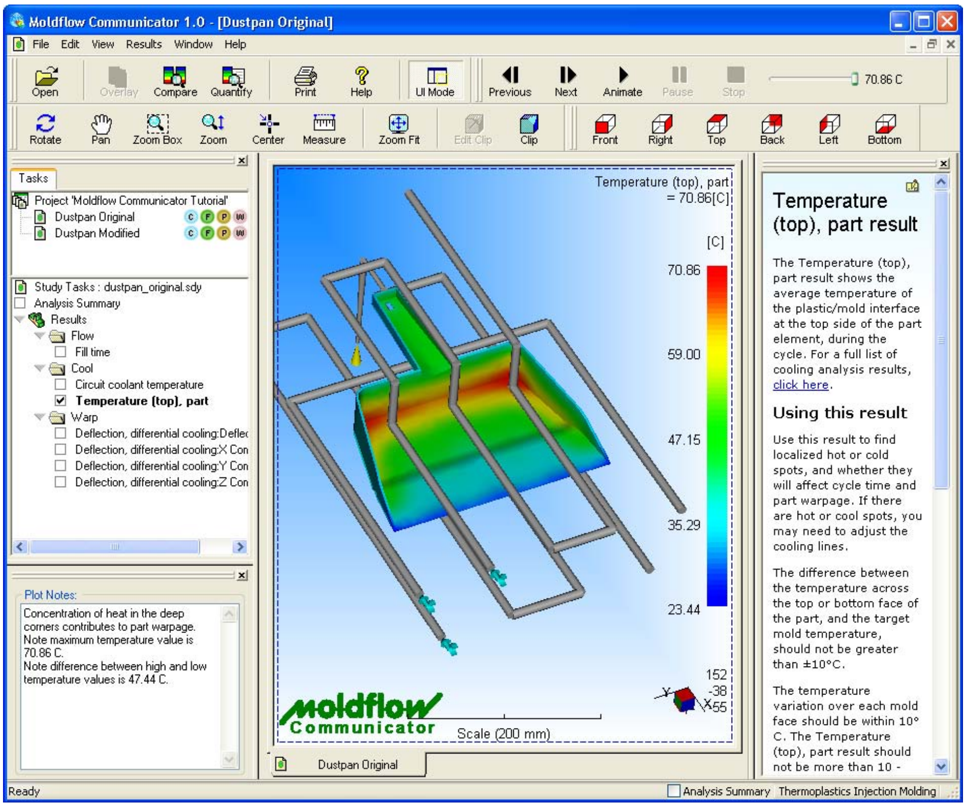


Figure 538: Moldflow Communicator interface

The components and functions of the interface are:

- Menus.
  - ➤ Pull down lists that provides quick access to the functionality within Moldflow Communicator.
- Toolbars.
  - ➤ Series of icons that allow quick access to most commands in Moldflow Communicator. Toolbars can be added and removed as necessary.
- Panel.
  - ➤ Area on the side of the screen that contains panels used for project and study (model) management.
- Tasks.
  - ➤ Project - lists the studies and analyses sequences performed.
  - ➤ Study task - displays the analysis inputs.
- Plot notes - contains any extra information added by the creator of the Results File.
- Display window - model and analysis summary is displayed in this area.
- Dynamic help - provides information related to the result displayed on the screen.

- Context menu.
  - A context sensitive menu accessed with a right click. Different options appear depending on where the cursor is when the context menu is activated.

You may turn on or off the different panes on the interface from the View menu as shown on Figure 539 below:

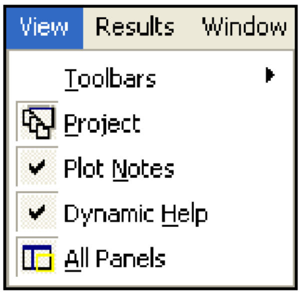


Figure 539: View menu options

Menus

The menus at the top of the Moldflow Communicator main window contain the commands available inside the program. The commands available in the menus depend on the stage of the reviewing results process you are currently in. Table 1 below describes the function of each menu.

Table 1: Moldflow Communicator menus and their functions

Menu	Function
File	Opening, closing, reviewing project properties, printing, setting system preferences and exit.
Edit	Copying images and saving images and animations.
View	Displaying/hiding the various windows, toolbars, etc. within Moldflow Communicator. Toggle for changing the icon size.
Results	Display options for visualizing results such as overlay, compare and quantify. Results such as fill time and air traps, fill time and weld lines, and fill time and orientation results (core/skin) can be overlaid.
Windows	Controlling the sub-windows in the display window.
Help	Accessing the Online Help and other information about Moldflow Communicator and Moldflow, including keyboard shortcuts, tutorials and connecting to Moldflow on the Web. Information about the program release and build number are also available from this menu.

Toolbars

Most of the commands available in the menu are also available on a toolbar. Each toolbar can be moved to any part of Moldflow Communicator by clicking on it and dragging. The toolbars can be displayed or hidden using the **View** ➡ **Toolbars** menu option. Table 2 below describes each toolbar and its function.

Table 2: Toolbars and their functions

Toolbar	Function
Standard	Opening, overlaying, comparing, quantifying, printing, online help and panels display.
Viewer	Manipulating the display of the model.
Animation	Controlling the animation of results.
Standard view	Selecting standard view rotations.

A complete list of icons and actions available are shown in Table 3 to Table 6.

Table 3: Standard toolbar icons

Icon	Name	Action
	Open	Opens a Moldflow Communicator file.
	Overlay	Allows for view more than one result at a time on the same model. Fill time and air traps, fill time and weld lines, and fill time and orientation results (core/skin) can be overlaid.
	Compare	Displays each study one above the other, and selects the first result from each. The panes are locked together, so changes made in one pane also appear in the other.
	Quantify	Compares all studies with a set of criteria contained in a criteria file (*.criteria), and presents the data as a table, highlighting results that do not match the criteria.
	Print	Used to display the **Print** dialog where you can view and modify the current print settings and then print the currently selected window.
	Help	Opens the Moldflow Communicator Help window.
	View All Panels	Shows/hides all panes in the main window. Use this to get a larger view of your model or results.

Table 4: Viewer toolbar icons

Icon	Name	Action
	Rotate	Dynamically rotates the part.
	Pan	Moves the part within the display.
	Zoom Box	Zooms in an area that you select by banding.
	Zoom	Increases/decreases the model magnification as you hold the left mouse button and drag the cursor up or down the screen.
	Center	Moves the location that you click on to center of the screen. This will become the new center of rotation.
	Measure	Measures the distance between two nodes or any other location, on the model. The part can be manipulated between picks.
	Zoom Fit	Re-sizes the model so the whole part fills the screen.
	Edit Clip	Open a dialog to move the active cutting plane. It can be moved manually or animated by entering an incremental value.
	Clip	Opens a dialog to define and activate a cutting plane to see inside the model.

Table 5: Animation toolbar icons

Icon	Name	Action
	Previous	Displays the previous frame of the animation.
	Next	Displays the next frame of the animation.
	Animate	Animates the result in time or thru the scale.

Table 5: Animation toolbar icons

Icon	Name	Action
	Pause	Temporarily stops the animation at its present view.
	Stop	Terminates the animation.
	Animation Control	Displays and controls the progress of the selected result animation.

Table 6: Standard views toolbar icons

Icon	Name	Action
	Front	Displays the front side of the model, Rotation 0 0 0.
	Right	Displays the right side of the model, Rotation 0 -90 0.
	Top	Displays the top side of the model, Rotation 90 0 0.
	Back	Displays the back side of the model, Rotation 0 180 0.
	Left	Displays the left side of the model, Rotation 0 90 0.
	Bottom	Displays the bottom side of the model, Rotation -90 0 0.

The buttons on the toolbars are not always available since the functions for the buttons might not apply to the selected object.

Panels

The Task panel displays the **Tasks** tab which is divided into two sections as shown in Figure 540:

- Project pane.
- Study tasks pane.

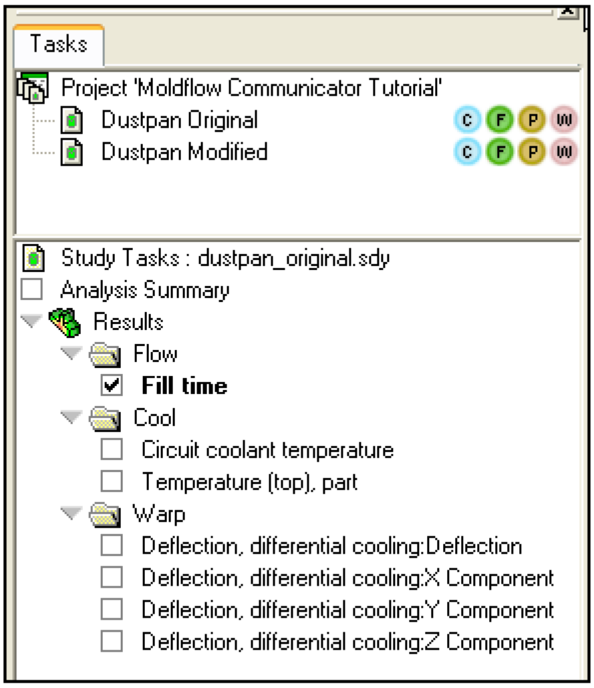


Figure 540: Tasks panel in Moldflow Communicator

A divider bar between the Project pane and the Study tasks pane is movable, so the relative sizes of the two panes can be changed.

Plot notes

This panel is used to display explanatory text associated with the selected result. Plot notes are created in MPI by the CAE analyst before the Moldflow Results File (*.mfr) was created. Plot notes can be used to convey specific information that may aid the interpretation of the displayed result.

Display window

You can have up to two studies open at one time in the display window. Each open study in the display window is called a document. The display window can be enlarged by using the Show/hide panels from the View menu as shown in Figure 541.

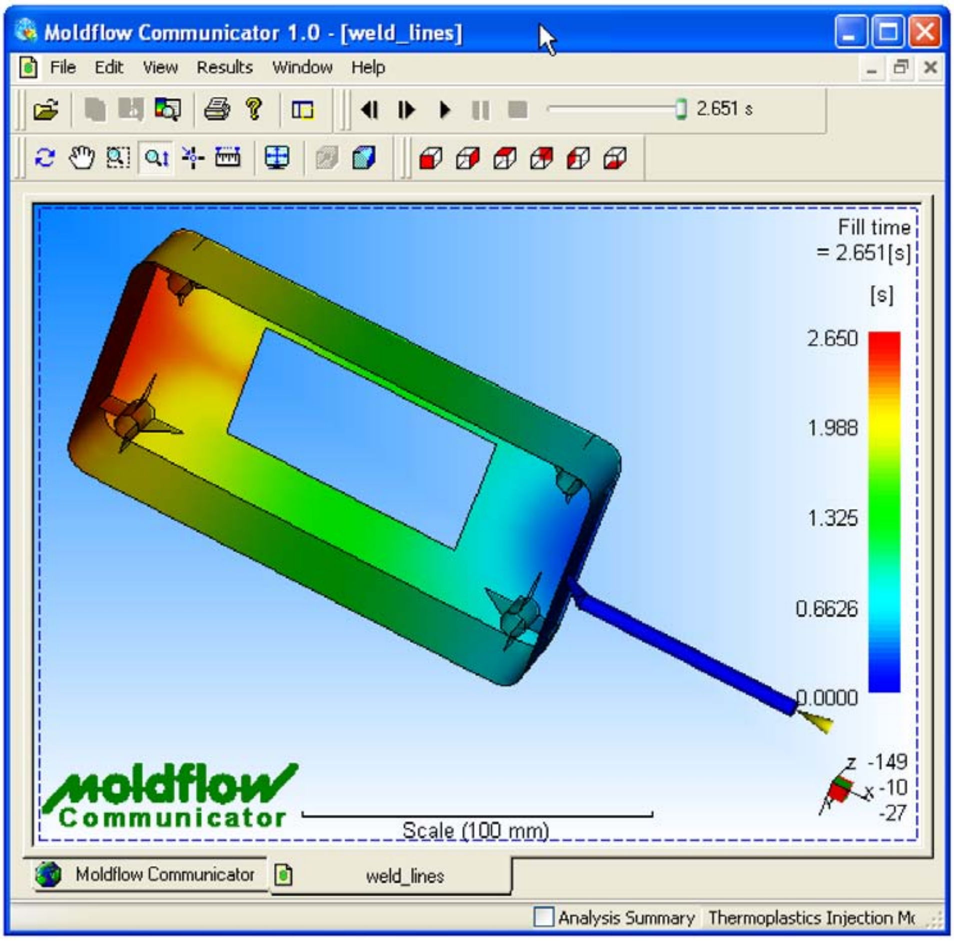


Figure 541: Moldflow Communicator display window - all panels hidden

Analysis summary

The analysis summary are text files. Their displayed is controlled by the checked box on the Study tasks, or at the bottom of the document window. Once they are displayed you can move from one log to another using the tabs on the top of the pages as shown in Figure 542.

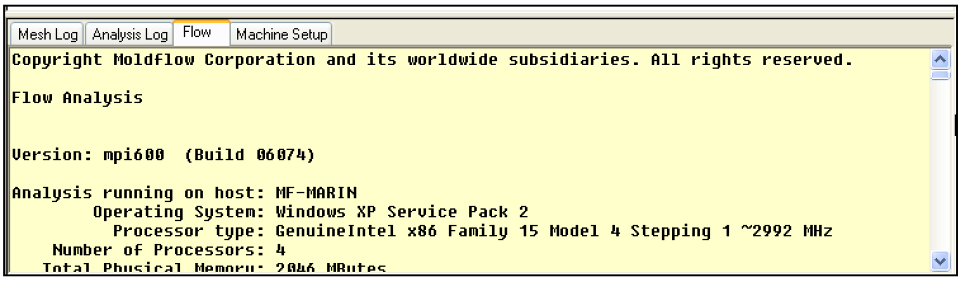


Figure 542: Analysis summary dialogs

Dynamic help

The Dynamic help panel automatically displays information associated with the currently selected result. Notice in Figure 543 the dynamic help panel is displaying information regarding the fill time result which is the result for the model on the display window. Dynamic help can be turned on and off from the View menu.

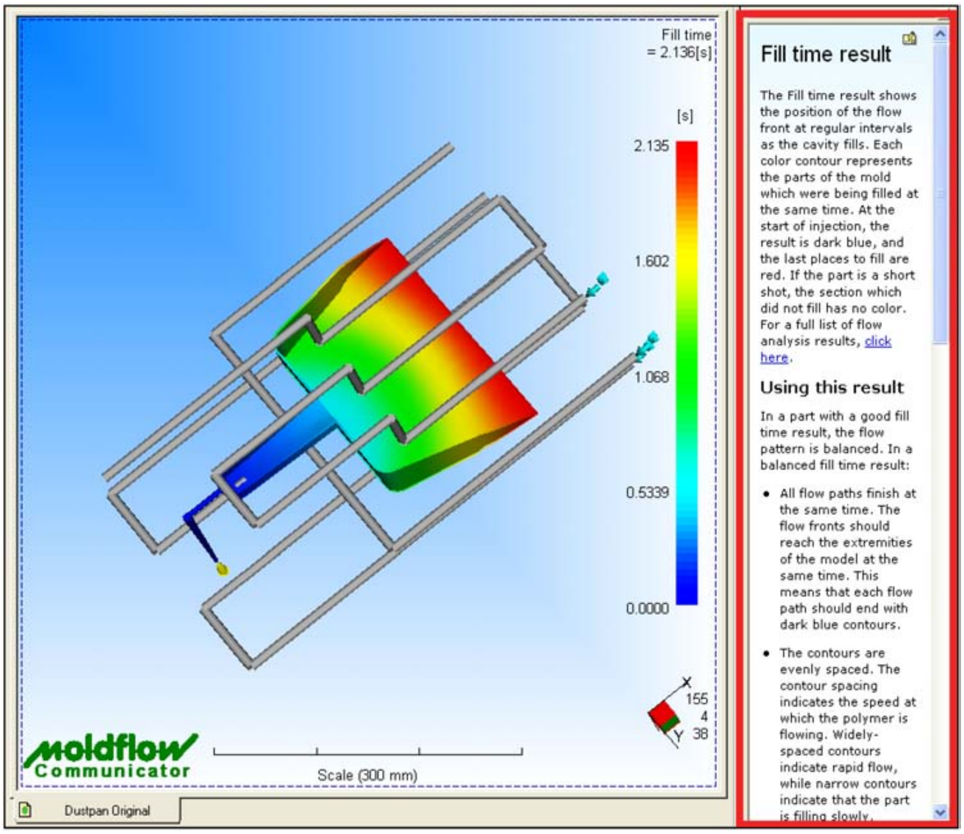


Figure 543: Dynamic help panel

Context menu

The context menu is a short menu that pops up after a right-mouse click. The context menu appears with different options, depending on what is highlighted or where the cursor is.

Different context menus appear at the following locations:

- The compare with criteria in the **Project pane.**
- Show on Analysis Summary in the **Study Tasks.**
- Hide or show on a **Result** listed in the Study Tasks.
- Toolbars display on a **Toolbar**.
- Viewer commands from the **Display window**.

The context menu often provides a very quick and handy way to access relevant functions of Moldflow Communicator. An example of a context menu is shown below in Figure 544.

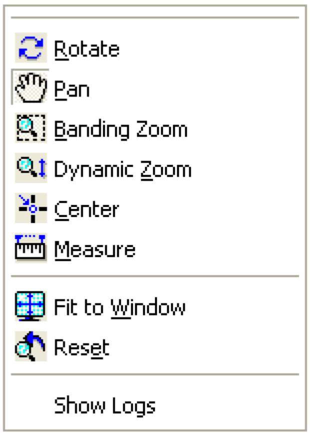


Figure 544: Context menu from the display window

Using Moldflow Communicator

Opening a results file

As soon as you open Moldflow Communicator the Launch wizard dialog appears, as shown in Figure 545 for you to choose or browse for a **Moldflow results file (mfr)** file.

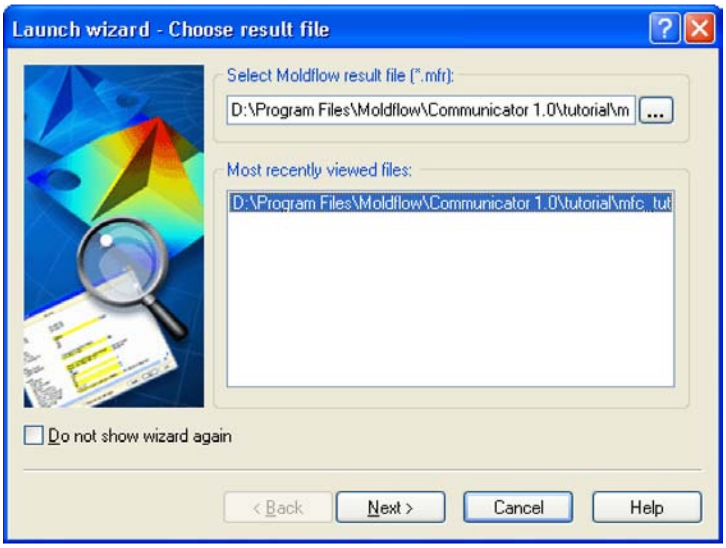


Figure 545: Launch wizard - Choose result file

You can also open the result file by clicking on the icon or click **File ➡ Open Project.**

If the Launch Wizard does not open when Moldflow Communicator is started, Click **File ➡ Preferences** and check **Use wizard at start**.

The Launch wizard offers three action options depending on what you would like to do with the results. The Launch wizard selection action page is displayed in Figure 546.

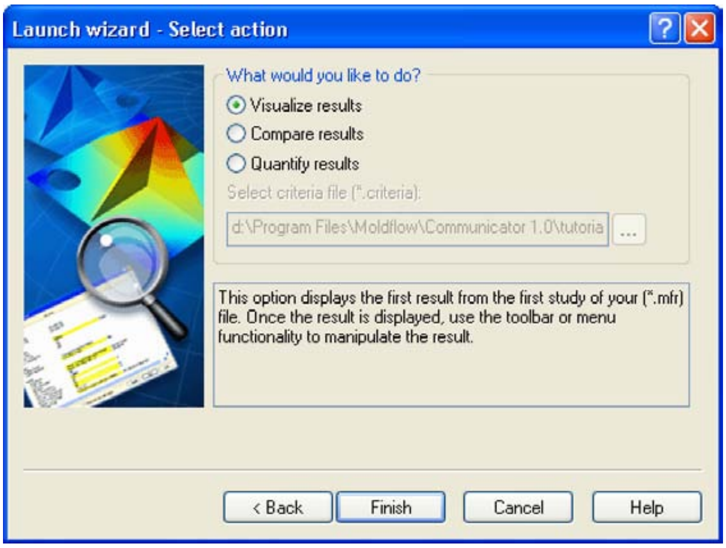


Figure 546: Launch wizard - Selection action

These action options details are:

- Visualize results.
  - This option displays the first result from the first study of your (*.mfr) file. Once the result is displayed, use the toolbar or menu functionality to manipulate the results.
- Compare results.
  - This option displays the first result from each study, one above the other. The panes are locked together so it is possible, for example, to run a Fill Time animation on both results at the same time. Once the result is displayed, use the toolbar or menu functionality to manipulate the results.
- Quantify results.
  - This option creates a tabular summary of the studies compared to the criteria entered in a criteria file. Where the input or result is different from the criteria or between the studies, the associated table cells are highlighted.

Manipulating results

When one or two studies are displayed, there are many tools that are used to manipulate the model and results. These tools are listed in:

- Table 4, "Viewer toolbar icons," on page 854.
- Table 5, "Animation toolbar icons," on page 854.
- Table 6, "Standard views toolbar icons," on page 855.

Click on the result, in the study tasks list you want to display and then used the tools necessary to view the results.

Clipping planes

The tools, Clip 

and Edit clip 

are used to activate and move clipping or cutting planes on the model. These can be used to remove parts of the model to see smaller features that would otherwise be difficult to see.

Clip

The **Clip** tool opens the dialog shown in Figure 547. With is dialog, you can turn on one of the pre-defined planes or create a new plane. The plane is defined by the screen. Click the **Make Active** button to make the highlighted layer active. Click the **Flip** button to change the direction of the clipped part of the model.

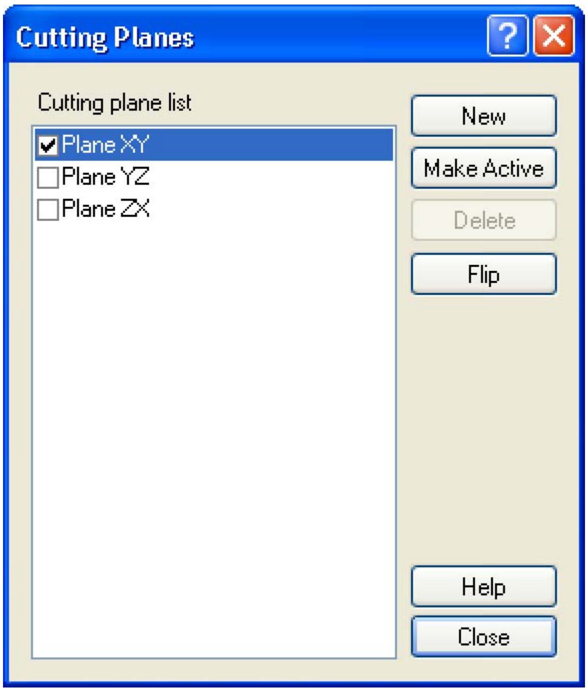


Figure 547: Clipping plane activation

The plot shown in Figure 548 shows a 3D part that has a clipping plane defined so the filling pattern in this feature can be seen better.

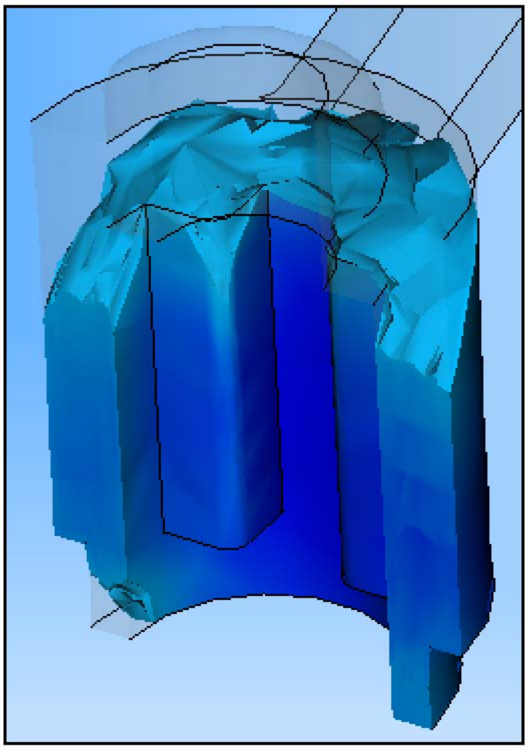


Figure 548: Clipped fill time plot

Edit clip

The **Edit Clip** tool opens the dialog shown in Figure 549. With this dialog open, drag the mouse up and down to move the clipping plane. Uncheck the **Show active plane** box to turn off the clipping plane shown by a transparent gray surface. Click the **Animation** button to move the clipping plane by the increments defined in the Distance field.

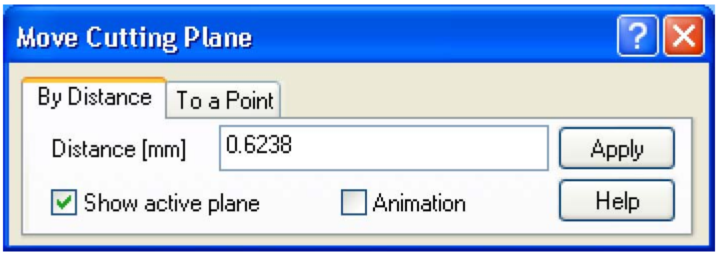


Figure 549: Moving a clipping plane

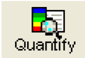
Comparing results

When the **Compare results** radio button in the Launch wizard or the **Compare** icon

or the command **Results** ➡ **Compare results** is used, both studies in the MFR file are opened, and tiled horizontally. Then any of the available results can be shown using the same tools described above.

Quantifying results

When the **Quantify results** radio button in the Launch wizard or the **Quantify** icon



or the command **Results ➡ Compare results** is used, a Select file dialog box is opened. The file required is a ***.criteria** file. It contains a long list of categories and fields listed in APPENDIX A **Criteria file contents** on page 29. The criteria file is produced in Moldflow Plastics Insight. Figure 550 shows an example of a comparison between two studies and quality criteria. Lines highlighted in yellow indicate where the studies differs from the criteria.

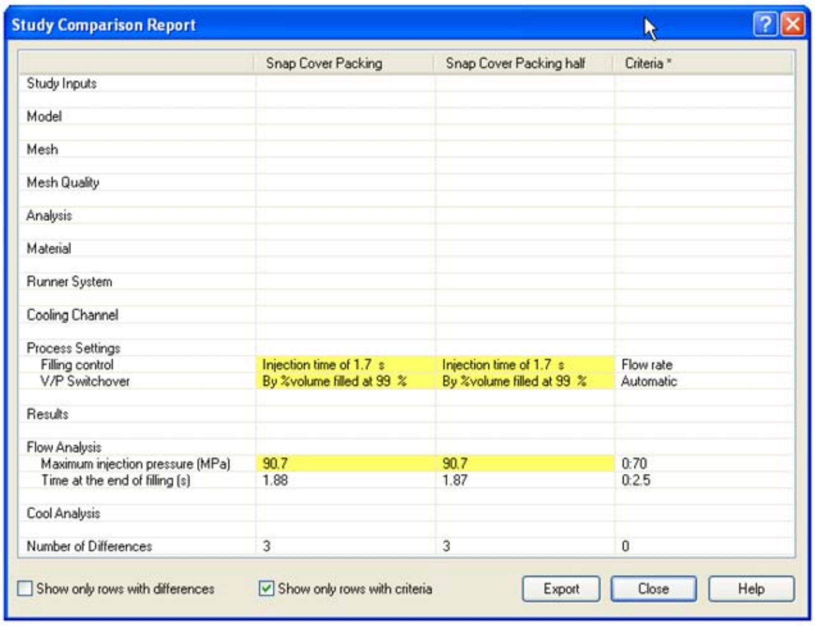


Figure 550: Study comparison report

Once the result file is opened you may toggle between **Compare Results** and **Quantify Results** at any time.

Results file contents

The Moldflow results file contains information to quantify the source of the results file and the quality of the data. This information comes in the following categories:

- Creator.
- Analysis product.
- Model attributes.
- Results Attributes.

Creator information

The command **File ➡ Project Properties** allows you to view information about the studies:

- Creator.
- Company.
- Contact information.

Analysis product information

Analysis product information states which solver being used, its release number and date of release. This information is stored in the Screen output file for every analysis. When a Moldflow results file is written, the Analysis Log files for every study are written as well. Analysis Log files are opened by the **Analysis Summary** check box at the bottom of the study window, as shown in Figure 551.

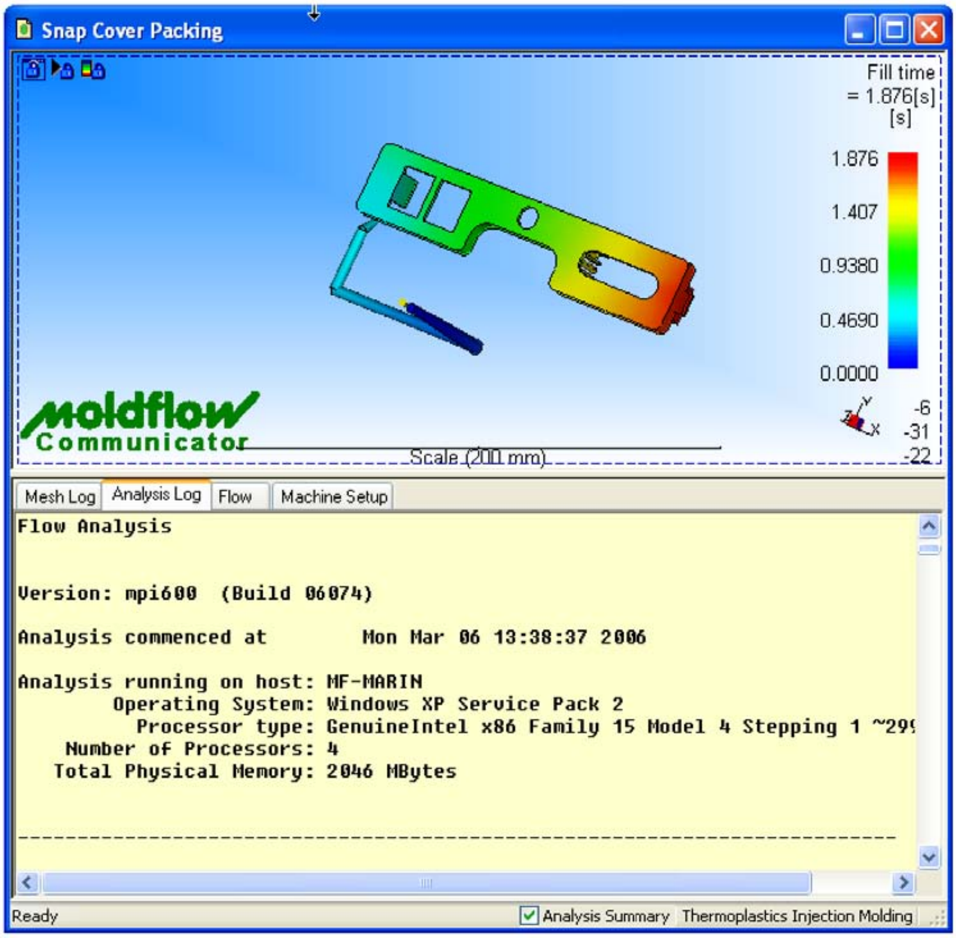


Figure 551: Log files in Moldflow Communicator

Model attributes

Model attributes include features of the model such as mesh quality, material model, etc. They can be seen when reviewing the study comparison report as shown in Figure 550. These attributes can be compared between models or defined quality criteria. Attributes are described in **Criteria file contents** on page 29.

Result attributes

Result attributes include features of a result such as projected area, minimum bulk temperature, fill time, etc. They can be seen when reviewing the study comparison report as shown in Figure 550. These attributes can be compared between models or defined quality criteria. Attributes are described in **Criteria file contents** on page 29.

Preparing files in Moldflow Plastics Insight

The file format called **Moldflow results file** (*.mfr) was created for use in Moldflow Communicator. Moldflow results files contain one or two studies, up to eight results from each study, plot notes and the screen output file.

> Moldflow Communicator does not have the ability to turn on and off layers or to change result properties. The necessary layers must be turned on and the result properties must be set as desired in MPI before creating the Moldflow result file.

It is possible to create multiple copies of a result with different plot properties, such as scaling, color and mesh display. In this way, you can have a general result alongside a scaled result that illustrates a point you want to make. Plot notes created in MPI are automatically displayed when the result is displayed.

Entering creator information

In MPI, the command **File ➧ Project Properties** allows you to enter information about the studies:

- Creator.
- Company.
- Contact information.

This information is automatically stored in the Moldflow results file, and is visible in a non-editable dialog in Moldflow Communicator, with the same command, **File ➧ Project Properties**.

Marking results to export

Moldflow results files are designed to be as small as possible by limiting the information that is in the file. One way this is done is to limit the results that are written to the Moldflow results file. Every result to be exported to the Moldflow results file much be marked for export in MPI.

To mark results for export to the Moldflow results file

1. Right click on the result that you want to export.
2. Select **Mark for export**, as shown in Figure 552.
   - An asterisk "*" is appended to the result name to indicate the result has been marked for export.

Only eight results can be marked for export into one Moldflow Results file.

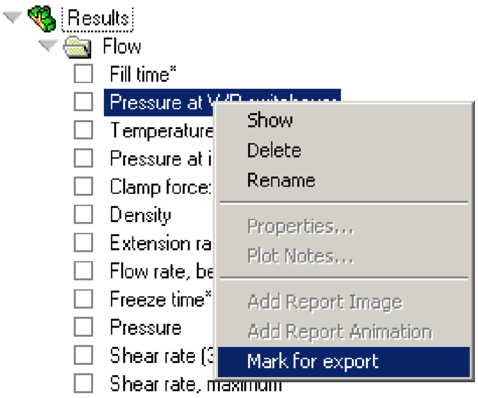


Figure 552: Marking Results for export

Creating plot notes

Plot notes available in Moldflow Communicator are created in MPI using one of the following methods to open the plot notes dialog:

- Right click on the result in the study tasks list and select **Plot Notes**.
- Click the Plot notes icon on the results toolbar.
- Click **Results** ➡ **Plot notes**.

The plot notes dialog will appear. Type the notes that correspond to the displayed results. Click **OK** to save the notes. Plot notes are included for the marked and exported result in the MFR file.

Creating the Moldflow results file

Once all the results are marked for export, and the all the appropriate layers are turned on, the studies must be saved. Once saved, the command **File** ➡ **Export** is used to create the Moldflow results file.

To create a Moldflow results file

1. Select the studies to exported in the Project view on the tasks pane in MPI.
2. Click **File** ➡ **Export**.
3. Navigate to the folder where the file is to be written.
4. Enter the result file name.

5. Ensure the file type is “Moldflow Results File”.
6. Click **Save**.

Creating the Analysis criteria file

A criteria file describes the values and ranges that attributes must have for the part to be considered of high quality. Values outside the desired range are highlighted in yellow when you compare a study with a criteria file.

To create a criteria file

1. Click **Tools ➡ Criteria editor...** to open the criteria editor dialog, shown in Figure 553.
2. Activate the check boxes beside the names of the quality criteria you would like included in the criteria file.
3. Enter a value for the attribute, or choose a value from the drop-down list. A range can be specified by using a colon (:).
4. Click **OK** to save the criteria file.
   - A dialog appears asking for a name for the new criteria file.

To edit an existing criteria file

1. Click **Tools ➡ Criteria editor...** to open the criteria editor dialog, shown in Figure 553.
2. Click the icon ... to read in an existing criteria file (*.criteria).
3. Navigate to the folder containing the file and open it.
4. Modify the names and values as necessary.
5. Click **OK** to save the criteria file.

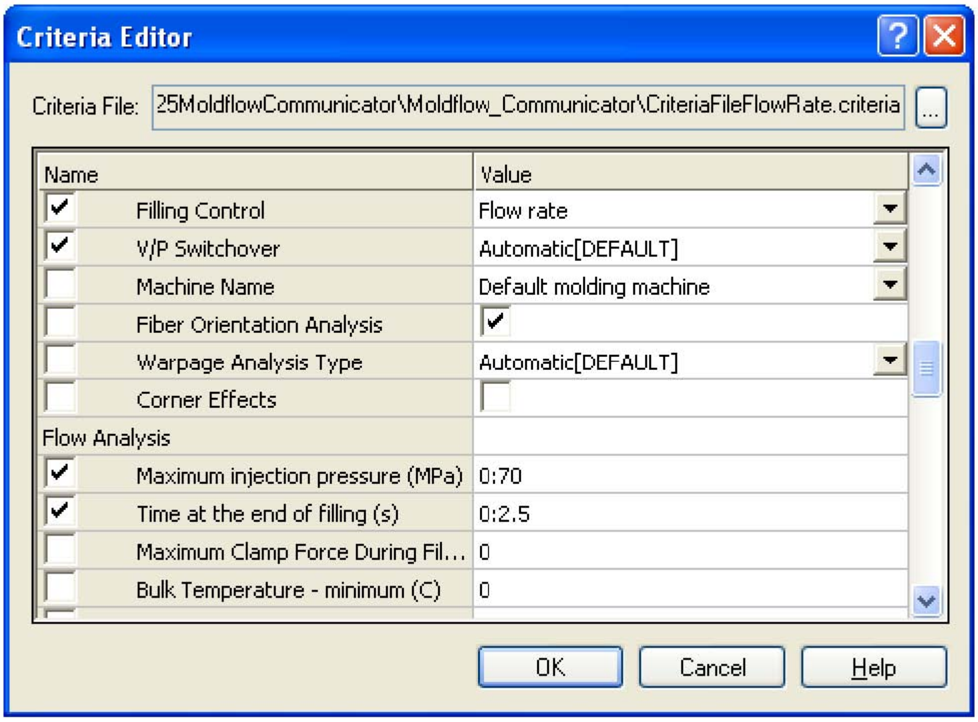


Figure 553: Criteria editor

What You've Learned

Moldflow Communicator is a free standalone product that is used to visualize Moldflow results. The interface is very similar to Synergy's. Moldflow Communicator reads in a new file format called Moldflow results file (MFR) and a criteria file. Both files are created in Synergy. The MRF can contain up to two studies and 8 results per study. The results are marked for export from the study tasks list and the **File ➡ Export** command is used to create the MRF.

CHAPTER 24

Job Manager

Aim

To learn about the various features of the Job Manager in order to manage analyses efficiently.

Why do it

Once the analysis is ready to be run, it is important to understand how the job manager works. You can save a lot of time and run each analysis more efficiently using some of the features of job manager.

Overview

The Job Manager dialog allows you to control the jobs for all 3 mesh types that are running, or are ready to be run. You can start running a job (analysis), allow a job to begin at a later time, change the priority of pending jobs, queue jobs, and configure a server machine to run jobs.

The Job Manager has the following sections:

- Studies.
- Jobs Preview.
- Submit to.
- Control tabs, including:
    - Add Server.
    - Remove Server.
    - Run/Pause batch queue.
    - Abort job.
    - Properties.

Theory and Concepts - Job Manager

Job Manager

The Job Manager is the tool that MPI uses to control everything related to running analyses and generating meshes.When you double-click the **Analyze Now** icon in the study tasks list, you are taking a shortcut for launching a job.You are sending an analysis job to the priority queue of the local host machine.

When any analysis is launched, the study file is copied and the analysis performed in a **working directory called mf1, mf2, mf3**, and so on. These are found in the temporary directory, on your local drive. The default temporary directory is **C:\MPI 6.0 Temporary Files**. Using these working directories allows the analysis to run using different disk resources to the actual project directory making the calculation process more efficient.

All the results files are copied back to the original project folder when the job is done. The Job Manager controls this process as well as multiple analysis servers, analysis queues, any jobs running or ready to be run; all are controlled within the Job Manager.

Opening the Job Manager

The Job Manager can be accessed by:

- Selecting the Job manager icon (not on a standard toolbar).
- Clicking **Analysis ➡ Job Manager.**
- Using the shortcut **CTRL + J.**

- Right-clicking the mouse on the **Analyze Now** icon in the Study Tasks pane and selecting Job Manager as shown in Figure 560.

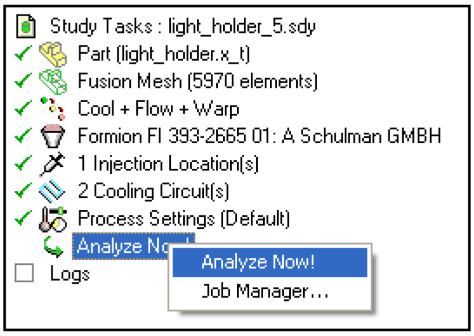


Figure 560: Study Task context menu

Job Manager Dialog

Figure 561 shows the Job Manager with several jobs being managed. Table 7 describes the features and functions on the Job Manager dialog.

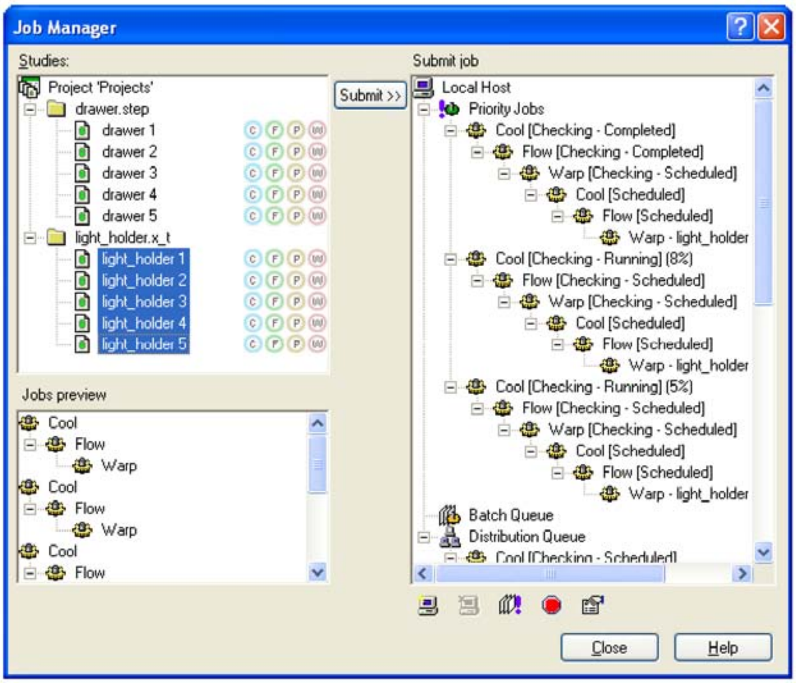


Figure 561: Job manager dialog

Table 7: Job manager features and functions

Feature/Function	Description
Studies pane:	The Studies pane shows the current project with its tree structure.This is the view as the Project View pane.From this pane you can see the studies that have and have not been run.You select the studies to be run from this list and assign it to the queue of your choosing.
Jobs preview pane:	The Jobs preview pane shows the analysis sequence of the highlighted study that still need to be run.
Submit job pane:	The Submit job pane shows the currently defined job servers, the available queues on each job server and any jobs currently running or assigned to each queue.
Add Server: 	The Add server icon allows you to add a job server for running jobs on a host machine.
Remove Server: 	The Remove server icon allows you to remove the selected server from the Job Manager.
Run/pause batch queue: 	The Run/pause batch queue icon allows you to start or stop any jobs in the batch queue. Pausing the batch queue will prevent any new jobs in the queue from starting. The currently running job will finish.
Abort: 	The Abort icon allows you to cancel the selected job.

Table 7: Job manager features and functions

Feature/Function	Description
Properties:	The Properties icon opens the Job server properties dialog.This allows you to setup and modify analysis servers.

Job Server

The **Job Server Properties** dialog shown in Figure 562 is used to setup and modify analysis servers.The features of the job server properties dialog are listed in Table 8:

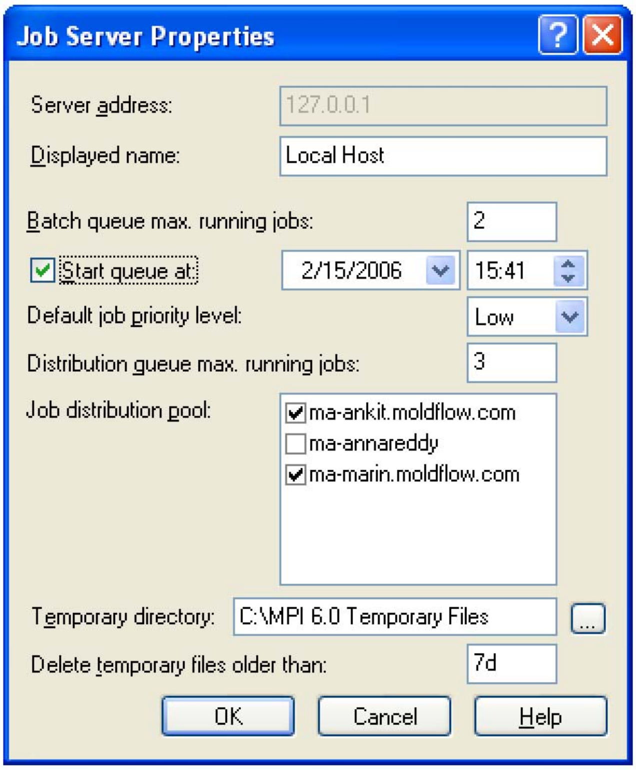


Figure 562: Configure server connection

It is not recommended to change the Temporary directory.

Table 8: Configure server connection dialog

Feature	Description
Server address:	Enter the address of the machine on which you want the analysis to run, it can be by the machine name or IP address. Normally, the machine name is entered.Do not use IP address if the new server doesn’t have static IP address set up.
Displayed name:	Enter the name of the server as you would like it to be displayed in the Submit job pane.The default is the same as the Server address.

Table 8: Configure server connection dialog

Feature	Description
Batch queue max. running jobs:	Enter the maximum number of batch jobs that you want to run at one time. The maximum number is equal to the number of processors on the machine.
Starts queue at:	Check this box to specify the date and time you will like the batch queue to start.
Default job priority level:	Set the processor priority that an analysis will take on the server. Low is the default setting and is the recommended level.If no other process needs the CPU the analysis will take 100% of the CPU.If another process needs the CPU, like moving the mouse, it will take priority.
Distribution queue max. running jobs:	Enter the maximum number of jobs that you want to run at one time on each server listed in the distribution pool. The maximum number is equal to the number of processors on the machine.
Job distribution pool:	Lists the available job servers, and indicates which of these job servers will be used by the Distribution Queue on the local machine. Jobs submitted to the pool are immediately broadcast to all servers in the pool. Each of the servers then broadcasts its own available CPU count. The job is submitted to the server with the most CPU's available. If there are no available CPU's across the whole pool, the job remains in the queue. The computers need to be on the same network and have the same release of MPI installed.
Temporary directory:	Sets the path to the temporary directory used by the Job Manager. The default path was set when MPI was installed.This shouldn't be changed. If you change the path, ensure that the drive that contains the temporary directory has sufficient storage space, especially when you run a cooling analysis.Editing the path manually can cause the analysis to fail.
Delete temporary files older than:	Set how many days the job manager temporary files are kept before they are deleted. The default is set to 7d (7 days).

Submit Job Pane

The **Submit** job pane in the Job Manager, shown in Figure 561 shows the jobs and host computers. Using this pane, analysis jobs can be added to an analysis queue and monitored to see their progress. Table 9 describes the analysis queues.

Table 9: Analysis queue descriptions

Queue	Description
Priority Jobs:	Priority jobs start immediately when placed into the queue.This is the queue used when you click Analyze now in the Study Tasks List.
Batch Queue:	Batch jobs are started when the Run batch queue button is clicked or at the set date and time as entered in the job server properties dialog. The batch queue is most often used to launch jobs at night.

Table 9: Analysis queue descriptions

Queue	Description
Distribution Queue:	Distribution jobs are sent to any server that has an available license and available CPU.The license can be floating or node locked. If the license is node locked, then it should be node locked to that server not the local computer.

Properties of Job Running

The properties of any analysis running can be examined at any time.

1. Open the Job Manager by selecting **Analysis ➡ Job Manager (CTRL + J)**.
2. Right click over the analysis of your interest, as shown in Figure 563.
3. Select **Properties** from the context menu. The program will list the following information, as shown in Figure 564:
   - Analysis type.
   - Model name.
   - Project directory
     - ➤ Location of the original study file.
   - User.
   - Submitted from
     - ➤ Host machine from where the analysis was launched.

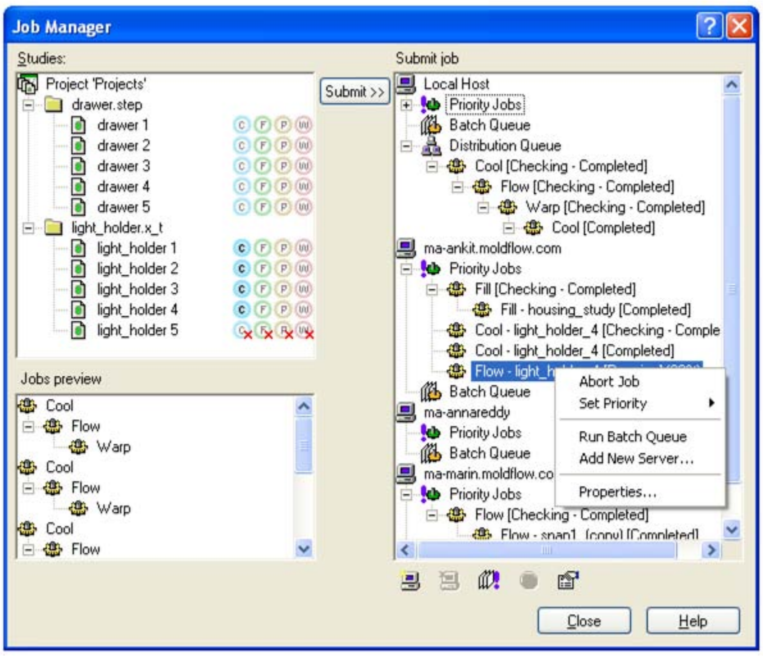


Figure 563: Analysis properties

Figure 564: Analysis properties dialog

Job Manager use examples

How to launch job to the priority queue

Before starting the analysis, ensure that all pre-processing tasks required to fulfill the analysis have been completed.

There are four ways to submit an analysis to a priority queue.The first three only submit to the local machine.The last method can submit an analysis to any server.

1. Double-click on the icon **Analyze Now!** icon in the study tasks list.
   - This is the most common method.
2. Right-click on the icon **Analyze Now!** icon in the study tasks list and select **Analyze Now**.
3. Click **Analysis** ➡ **Analyze Now**.
4. Right-click on the icon **Analyze Now!** icon and select **Job Manager**.
   4.1. Select the study to be launched in the **Studies** pane.
   4.2. Drag the study to the priority queue on any active server.

The Job Manager can be opened with **CTRL + J** or **Analysis** ➡ **Job Manager**.

How to send a job to any queue

1. Right-click on the icon **Analyze Now!** icon and select **Job Manager**.
2. Select the study or studies to be analyzed.
3. Click on desired queue on any available server, in the **Submit job** pane.
4. Click the **Submit >>** button.

The selected studies can be sent to a queue by dragging them as well.

How to stop (abort) an analysis

Sometimes an analysis that is running must be aborted.This is normally because the analysis has some input that is not correct.

1. Select **Analysis** ➡ **Job Manager** from the menu.
2. Click on the analysis to be stopped, in the **Submit job** pane.
3. Click the **Abort** icon  at the bottom right of the Job Manager window.

How to start the batch queue

Once one or more studies are submitted to the batch queue, the jobs must be started. You can use a manual or an automatic method.

Manual batch queue starting

1. Click **Analysis** ➡ **Job Manager**.
2. Click the **Run/pause batch queue** icon  to start the batch queue.

Automatic batch queue starting

With this method, the date and time for the batch queue to start must be defined.When this time is reached, the batch queue will automatically start.

1. Select **Analysis** ➡ **Job Manager** from the menu.
2. Highlight the server the batch queue belongs to.
3. Click the **Properties** icon.
4. Check the **Start Queue at:** box.
5. Enter the **date** and **time** that you want the batch queue to start.
6. Click **OK**.

Running Jobs without the Job Manager

Analysis jobs can be run without using the Job Manager.This is often done on hardware platforms such as Linux on which Synergy (MPI user interface) is not supported.It can also be used as a way to automate analysis execution outside of Synergy.A utility called **runstudy** is launched from a command line to run jobs without using Synergy or the job manager.

Using runstudy

PC

1. In MPI/Synergy, set up and save the analysis you want to run.
2. Open a **DOS** prompt by selecting **Start** ➡ **Programs** ➡ **Moldflow Plastics Insight <release_no>** ➡ **Plastics Insight Command Shell**
   - Where <release_no> is the version number of the MPI release being used.
3. Use the DOS **cd** command to navigate to the folder where the study you want to launch is located.
4. Enter **runstudy** on the command line without any parameters to see the help for the command.Figure 565 shows the output from the utility **runstudy** showing the help available for the utility.

Unix

Start a UNIX shell and enter **runstudy.**Executing runstudy is the same for all systems.

```
C:\My MPI 6.0 Projects>runstudy
--------------------------------------------------------------
NAME:
   runstudy - Command line launching of Moldflow analyses
--------------------------------------------------------------
SYNOPSIS:
runstudy
    [-help]
    [-project project_file]
    [-temp temp_dir]
    [-keeptmp]
    study_name
--------------------------------------------------------------
DESCRIPTION:
   study_name               study file name
   -project project_file    to specify that the study name
                            specified should be looked up in
                            the specified project file.
   -help                    to print this help message.
   -temp temp_dir           to specify the temp area to be used
                            (default is <current_dir>\temp).
   -keeptmp                 to specify that any temporary files
                            created should be retained.
------------------------------------------------------------
```

Figure 565: Runstudy help from a DOS prompt

Examples

`runstudy -project bracket.mpi two_gate.sdy`
The above example launches the **two_gate** study in the **bracket** project in the current directory. The analysis will store its temporary files in a subdirectory called <current_dir>\temp, the default location.
`runstudy -project bracket two_gate`

The above example launches the project and study without the default file extensions.
runstudy -project bracket two_gate -temp c:\temp two_gate
The above example launches the analysis with a specified temporary directory location.

What You've Learned

Using the Job Manager is easy.From the Job Manager all analyses can be launched, monitored or edited on all servers.

The utility called **runstudy** is used to launch analyses without using the job manager window from the Synergy interface.

CHAPTER 25

Guided Project

Aim

The aim of this chapter is to work through an entire project; from receiving a CAD model and running different analysis, to finding and solving the problems associated with the part.

Why do it

By doing this in-depth practice of the entire analysis process, you will be much more confident in how the entire analysis process works, and will be able to apply the analysis procedure to your own part designs. This practice includes using the mesh tools to fix many kinds of model problems, using various analyses to identify filling-related problems, interpreting results to understand the problems, and test corrective actions to find a resolution. During the course of the chapter, you will employ Moldflow design principles to understand and solve the analysis problems.

Overview

This is a quick look at the procedures involved in this project which include:

1. Understand the design criteria.
2. Import the model.
3. Check the model for mesh errors.
4. Fix the mesh errors.
5. Run a gate location analysis.
6. Run a molding window analysis.
7. Run a fill analysis.
8. Interpret the results to identify problems.
9. Determine possible solutions.
10. Change the model or process settings to fix the problem.
11. Run an analysis to verify the problem is fixed.
12. Create a runner system for the part.
13. Analyze the initial runner system design.
14. Use flow analysis to optimize the runner design.

Theory and Concepts - Guided Project

Design questions to be addressed

In the part design process, there are a lot of questions and problems that need to be overcome before a successful part can be created. The training chapter Flow Analysis Steps contains a comprehensive listing and can be referred to for a broader overview, however, in this chapter, a good mesh will be created, and then an answer needs to be determined, step by step, for each of the following design questions:

- What is the best injection location?
- What are the optimum process settings for the part?
- What is the minimum diameter of the gate?
- What are the best part thicknesses?
- What are the correct runner dimensions?
- How long is the cycle time?
- What injection molding machine size is required?

You will not be able to determine all of the answers to the questions listed above in one analysis; therefore this chapter you will run the following analyses:

- Best Gate Location.
- Molding Window.
- Fill.
- Flow.

Preparing a finite element mesh

The specific steps for creating meshes for analysis in MPI are listed below. Figure 566 shows these steps in a flow chart. These steps are described in detail below.

- Prepare CAD Model.
- Import CAD Model.
- Set Mesh Densities.
- Generate mesh.
- Evaluate mesh.
- Cleanup mesh.
- Verify mesh is clean.

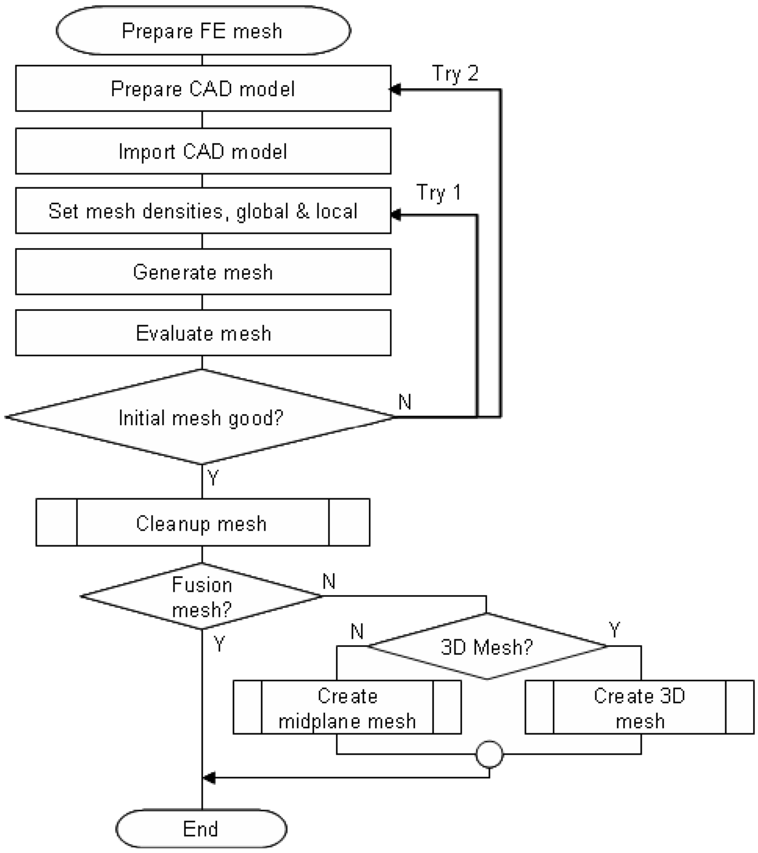


Figure 566: Preparing a finite element mesh flow chart

Prepare the CAD model

Most models analyzed in MPI use a CAD model as their source. This geometry must be free of errors. The most common file formats that are imported is IGES, and STL. Other formats can be imported, including:

- Solidworks 2005 part files (.sldprt).

- CatiaV5 part files (.CATPart).
- Pro/Engineer part files (.prt).
- Parasolid (.x_t, .x_b, .xmt_xmb, .xmb, .xmt).
- STEP AP203 (.stp, step).
- IDEAS Universal, mesh only (.unv).
- ANSYS Prep 7 (.ans).
- Nastran (.nas).
- Nastran Bulk Data (.bdf).
- Patran (.pat).
- Fem (.fem).
- Moldflow (.mfl).
- C-Mold (.cmf).

Some of the formats above require an import utility called Moldflow Design Link, (MDL).

Import CAD Model

Importing a model is just like a **File ➡ Open** command in most business applications such as a word processor. By default, the Import dialog will show all importable CAD file formats. The **Files of type** box allows you to specify the type of file you are looking for.

Set Mesh Densities

The mesh density is primarily controlled by the edge length of the element. The edge length is set globally. For file formats imported other than STL, the mesh density can be assigned locally by the NERBS surface.

In addition to the edge length, the mesh density is controlled by chord height on curved features. Occasionally, curvature control and proximity control are used to mesh curved and small features with a grading to flat large areas. The curvature and proximity controls are not always needed.

In many cases, a part is meshed with several different settings to determine the one to use.

Generate mesh

Once the mesh settings are set, the mesh generator is started to create the mesh.

Evaluate mesh

The mesh is evaluated several ways. If the mesh is not found to be acceptable, the part is re-meshed. In some cases, a new CAD model is imported. The mesh is evaluated by:

1. Visual inspection.
   - Looking at the mesh is the best first step to determine if the mesh is acceptable. Determine if the mesh looks too coarse or fine. Look for any problems. If problems are found, decide if the mesh should be created at finer density to fix the problem.
2. Mesh Statistics.
   - If initial visual inspection looks good, run Mesh Statistics on the part. This will find specific problems with the part.
3. Mesh Diagnostics.
   - Run a mesh diagnostic for any mesh quality attribute the mesh statistics flagged as a problem.
4. Inspect the thickness diagnostic.
   - Ensure the thickness of the part is correctly represented. If large areas of the part are not properly represented, remesh the part at a finer density.
5. Inspect the match ratio.
   - Ensure the match ratio does not have any areas that are wrong. If there are, remesh at a finer density.

Cleanup mesh

After the part has meshed, and evaluated using visual inspection, mesh statistics, and diagnostics, problems in the model need to be fixed, so a clean model is used for analysis, as shown in Figure 567. The first step is to use a Mesh Repair Wizard then if any remaining issues remain, a manual cleanup process is used.

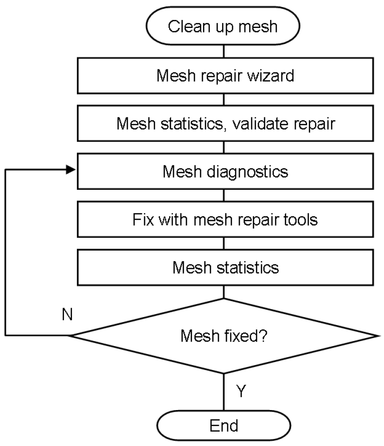


Figure 567: Clean up mesh flow chart

Mesh Repair Wizard

The **Mesh Repair Wizard**, is a tool that automatically cleans up most of the problem areas on your meshed model. The tool is opened with the command **Mesh ➡ Mesh Repair Wizard**, or on the **Mesh Manipulation** toolbar. The Mesh Repair Wizard includes the following pages:

- Stitch Free Edges.
- Fill Hole.
- Overhang.
- Degenerate Elements.
- Flip Normal.
- Fix Overlap.
- Collapsed faces.
- Aspect Ratio.
- Summary.

Using the Mesh Repair Wizard

A diagnostic for a given page calculates when you open the Wizard or move to the next page. If a problem is detected, a message appears. There are two check boxes:

- Show diagnostics.
    - This will toggle a diagnostic display for the mesh problem found. By default it is deselected.
- Show model.
    - This will toggle the meshed part. By default it is selected.

Fixing a problem

There are two choices for fixing a problem found on any of the wizard pages, including:

- **The Next button** - The Next button will fix the problem from the current page, then go to the next page of the wizard.
- **The Fix button -** The Fix button will fix the problem, and stay on the same page. This allows you to:
    - Review the number of changes made.
    - Review the model and undo the changes if necessary.
    - Change a setting and run the fix again.

If you do not wish to fix any problems on the page, click the **Skip** button.

Degenerate Elements

Most of the pages of the Mesh Repair Wizard, the default tolerances should be used. However, for the Degenerate elements page, it is useful to change the default tolerances. Increase the tolerance until some elements are below the tolerance. Use the dynamic navigator to investigate the problem areas. By increasing the tolerance, many higher aspect ratio elements are eliminated. Be sure to check the part after it has fixed elements to ensure the geometry is still correct.

Fix the mesh with mesh repair tools

In many cases, the Mesh Repair Wizard will fix most problems but not all problems. This is practically true for Midplane and Fusion. Manual mesh cleanup must be done to finish the process. The Tools pane contains the toolbox with all the tools in it. The list of tools related to mesh cleanup are listed in Table 10.

Table 10: Toolbox commands used for manual mesh cleanup

Icon	Tool	Commands	Icon
	Create/Beam/Tri/Tetra	Create Triangles	
		Create Beams	
		Create Tetras	
	Nodal Mesh Tools	Insert Node	
		Move Node	
		Align Node	
		Purge Node	
		Match Node	
		Merge Node	
	Edge Mesh Tools	Swap Edge	
		Stitch Edge	
		Fill Hole	

Table 10: Toolbox commands used for manual mesh cleanup

Icon	Tool	Commands	Icon
	Global Mesh Tools	Remesh Area	
		Smooth Nodes	
		Orient Element	
		Delete Elements	
		Project Mesh	
		Global Merge	
		Auto Repair	
		Fix Aspect Ratio	
		Create Regions	
		Orient All	

Verify mesh is clean

Once manual repairs are done, the mesh should be clean and ready for analysis. Use mesh statistics to verify the mesh is clean. There may be a point where cleaning the mesh any more is a waste of time. The mesh problems left to fix will take a long time in relation to the potential problems in the analysis.

Decide when to stop fixing

ALL intersections, overlaps, free and non-manifold edges, zero area elements, plus orientation **must** be fixed. The aspect ratio should be below 6:1, but if there are a few scattered elements in non-critical areas, slightly higher values are acceptable for Flow. If a cool and warp analysis is required, the mesh must be very good.

Clean up the layers

Normally during the process of mesh cleanup extra layers get created and used. After cleanup remove any extra layers and put the elements and nodes back in their original layers. One possible way is to:

- Make the Triangles layer active.
- Highlight all the layers to be deleted one at a time, and click the Delete button. Say **Yes** to move the entities to the active layer.

There will probably be nodes on the Triangles layer. To remove the nodes use the **Select By** command. Pick nodes to select, highlight the nodes layer and click Assign.

How to fix common problems manually

When automatic fixing methods do not fix all the problems and manual methods are needed, use the mesh diagnostics to display and find the problem areas. Table 11 summarizes the tools necessary to fix many common mesh problems.

Table 11: Methods for fixing mesh problems

Problem	Possible solutions
Low mesh match ratio	**1.** Decrease global or local edge length and mesh again.
Thickness	**1.** Increase the mesh density. **2.** If the mesh density is generally good, re-assigning the thickness, after other problems fixed. **3.** **Note:** do not change the thickness much. A large change will cause problems for Cooling and Warp.
Connectivity	**1.** Stitch free edge. **2.** Global merge. **3.** Merge node.
Intersections and overlaps	**1.** Try **Auto Repair** first. Check to make sure the auto repair did not create any problems. **1.** Merge nodes. **2.** Delete elements. then fill hole.
Free or Non-manifold edge	**1.** Create nodes. **2.** Merge nodes. **3.** Delete entities (elements). **4.** Fill hole.
High aspect ratio	**1.** Merge nodes. **2.** Swap edge. **3.** Insert node. **4.** Move node. **5.** Align nodes.
Un-oriented elements	**1.** Orient all. **2.** Orient element.

Generally meshing the part with a smaller global or local edge length will go a long way in fixing many mesh problems. It then becomes a compromise between computer time for each analysis and initial preparation time.

What You've Learned

In the analysis process, one of the most important parts of the process is getting the model ready for analysis. This steps include:

- Prepare CAD Model.
- Import CAD Model.
- Set Mesh Densities.
- Generate mesh.
- Evaluate mesh.
- Cleanup mesh.
    - Mesh repair wizard to find and fix most problems.
    - Run the mesh statistics to find remaining problems.
    - Fix problems manually with mesh repair tools.
- Verify mesh is clean.
    - Run mesh statistics to verify problems are fixed to an acceptable level.

A filling analysis can be used to answer many questions, including:

- What is the best injection location?
- What are the optimum process settings for the part?
- What is the minimum diameter of the gate?
- What are the best part thicknesses?
- What are the correct runner dimensions?
- How long is the cycle time?
- What injection molding machine size is required?

APPENDIX A

Thermoplastics Overview

Aim

Review important concepts regarding polymers (molding materials).

Why do it

By understanding how properties and characteristics of polymers affect the processing and final product you may interpret better the inputs and the analysis results.

Overview

In this chapter you will review information regarding:

- Polymers' definition & classification.
- Polymers' properties of interest.
- Thermoplastic material families & abbreviations.

Theory and Concepts - Thermoplastics Overview

What is a polymer?

The word “polymer” is a combination of the Greek words: “poly” which means “many” and “mer” which means “part”.

Structure of polymers

The basic structure of a polymer molecule can be visualized as a long chain of repeating units, with additional chemical groups forming pendant branches along the primary “backbone” of the molecule, as shown in Figure 602. Although the term plastics has been used loosely as a synonym for polymer and resin, plastics generally represent polymeric compounds that are formulated with plasticizers, stabilizers, fillers, and other additives for purposes of processability and performance. Other polymeric systems include rubbers, fibers, adhesives, and surface coatings.

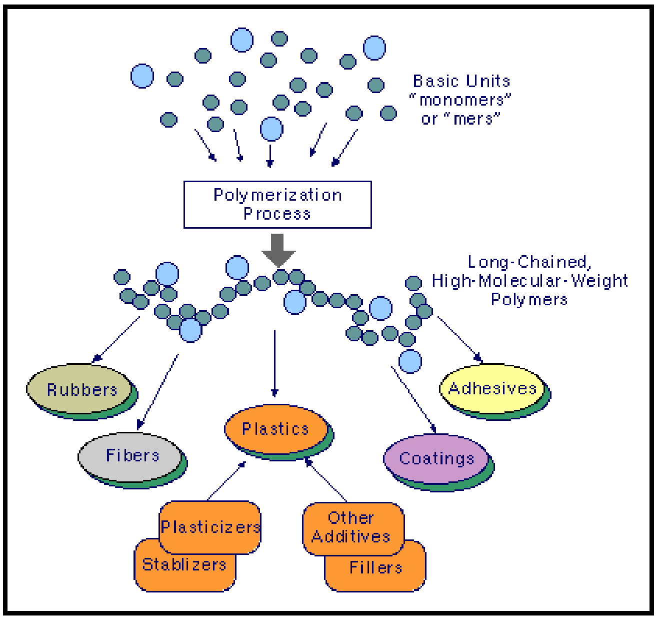


Figure 602: Polymer Family, the formation of plastics, and the polymerization process

Additives, fillers, plasticizers and stabilizers are components added which relate directly to properties processing and performance.

Polymers’ Classification

Based on the type of chemical reaction (polymerization) that links the molecules together, polymers are classified as either thermoplastics or thermosets.

Thermoplastics

The word "thermoplastic" is a combination of the Greek words: "therm" which means "heat" and "*plastikos*" which means "capable of being molded". Once the polymer is formed it can be heated and reformed over and over again (allows recycling).

Thermosets

The word "thermoset" is a combination of the Greek words: "therm" which means "heat" and "sets" irreversibly when heated. Therefore, can not be remelted. Once these polymers are formed, reheating will cause the material to degrade.

Thermoplastics classification based on morphology

In addition to the broad categories of thermoplastics and thermosets, thermoplastics can be further categorized into amorphous, (semi-)crystalline, or liquid crystal polymers (LCPs), depending on the polymer chain conformation or morphology. The microstructures of these plastics and the effects of heating and cooling on the microstructures are shown inFigure 603.

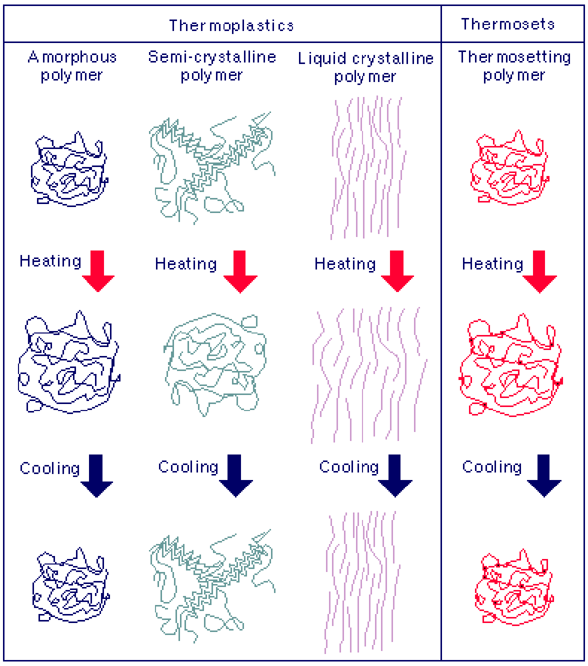


Figure 603: Microstructure of various polymers and the effect of heating and cooling during processing

Other classes include elastomers, copolymers, compounds, commodity resins, and engineering resins.

Properties of Interest

The analysis require accurate material-property data to generate the best predictions. The properties used to characterize the resins inside Moldflow programs are:

- Processing conditions:
  - Mold temperature.
  - Melt temperature.
  - Ejection temperature.
  - Maximum shear stress.
  - Maximum shear rate.
- Rheological properties:
  - Viscosity.
  - Juncture loss method coefficients.
  - Transition temperature.
  - Melt flow rate.
- Thermal properties:
  - Specific heat.
  - Thermal conductivity.
- Physical properties:
  - Melt density.
  - Solid density.
  - Specific volume (pvT diagram).
- Mechanical properties:
  - Elastic modulus.
  - Poisson's ratio.
  - Shear modulus.
  - Transversely isotropic coefficient of thermal expansion data.
- Shrinkage properties:
  - Shrinkage model.
  - Observed nominal shrinkage.
  - Observed shrinkage.

Processing conditions

The processing conditions are the range of values typically recommended by the resin manufacture to be used with the specific grade.

Mold temperature

This value corresponds to the temperature of the mold where the resin touches the mold.

Melt temperature

This value is the temperature of the resin, or melt, as it starts to flow into the cavity.

Ejection temperature

Is the temperature at which a material is rigid enough to withstand ejection. This value is used to determine the frozen layer fraction and the cooling time when set to automatic.

Maximum shear stress

This is the maximum shear stress for the material beyond which degradation starts to occur. You may use this value to optimize the size of the gates.

Maximum shear rate

This is the maximum shear stress for the material beyond which material degradation starts to occur.

Rheological properties

Polymer rheology is the most important property used in flow simulations.

Viscosity (η)

Viscosity is the polymer's resistance to flow. It is the ratio of the shear stress to the shear rate. It has the units of Pa.s (pascal.seconds) or poise. Typically it is measured using a capillary rheometer. The data is fit to form scientific equations which represent the flow characteristics. Viscosity is effected by:

- Shear rate, the melt flows easier when sheared.
- Temperature, the viscosity decreases when temperature is increased.

Most polymers exhibit two regimes of flow behavior, Newtonian and shear-thinning. Newtonian flow occurs at low shear rates, but with the increase of shear the viscosity tends to fall away in what is termed shear-thinning behavior as shown in Figure 604.

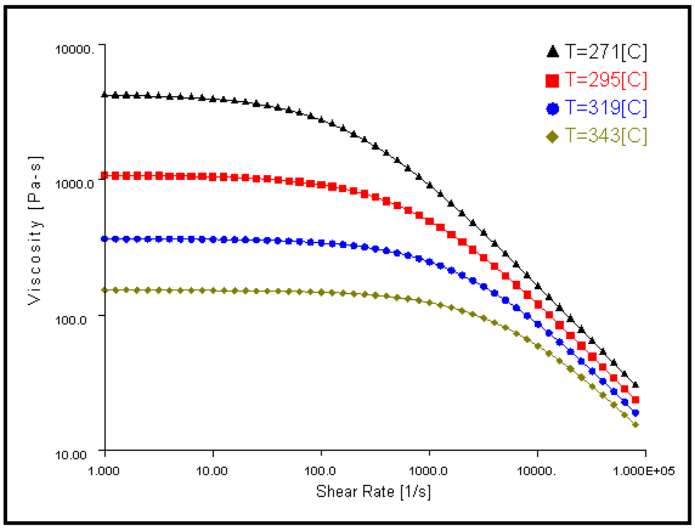


Figure 604: Viscosity curves

Juncture loss method coefficients

The juncture loss method coefficients consists of the calculation of the hydraulic loss that occurs when the melt passes through a large change in path diameter, such as from the end of the runner to the gate.

Transition temperature

The transition temperature is the temperature to where the polymer freezes, the melt-to-solid transition occurs.

- The transition temperature corresponds to the glass-transition temperature (Tg) for amorphous materials.
- The transition temperature is the crystallization temperature (Tc) for semi-crystalline polymers.

Melt flow rate (MFR)

The melt flow rate (MFR) is the industry standard which measure how easily the melt flows. For polyethylene the MFR is also known as Melt Flow Index (MFI) or Melt Index (MI). It is measured in grams per 10 minutes (g/10 min.). The problem with this value is that corresponds to a single data point at low shear rates.

Thermal properties

The heat flux calculation is essential for the packing and cooling analysis. The main thermal properties that are taking into account are discussed below:

Specific heat (Cp)

The specific heat is the ability of a material to hold heat. Is also a measure of a material's ability to convert heat input to an actual temperature increase.

Thermal conductivity (k)

The thermal conductivity is known as the ability of a material to conduct heat. It is also a measure of the rate at which a material can dissipate heat.

Thermal diffusivity (α)

The thermal diffusivity is the ratio of thermal conductivity to heat capacity expressed with the equation below:

$$\alpha = \frac{k}{\rho Cp}$$

Physical properties

Melt density (ρ_m)

The melt density is the density of the selected material in the melt state.

Solid density (ρ_s)

The solid density is the density of the selected material in the solid state.

Specific volume (pvT diagram)

The specific volume, also known as pvT diagram, describes the specific volume change as a function of temperature and pressure for polymers over the entire processing range. Notice how the specific volume decreases with the increase of pressure in Figure 605.

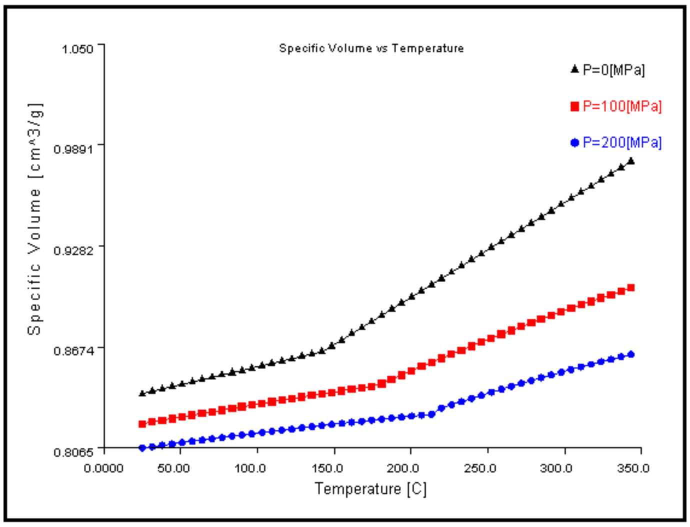


Figure 605: pvT diagram

The pvT diagrams are different for crystalline and amorphous materials as shown below:

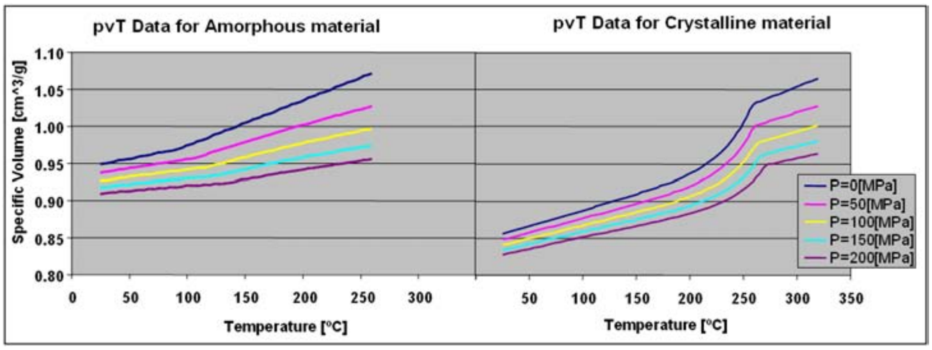


Figure 606: pvT diagram for an amorphous & crystalline polymer

Mechanical properties

For the warpage analysis additional properties are required.

Elastic modulus (E)

The elastic modulus is the force you need to provide to elongate the material.

Shear modulus (G)

The shear modulus (G) is the ratio of shearing stress τ to shearing strain γ within the proportional limit of a material. The shear modulus can be expressed with the following equation:

$$G = \frac{\tau}{\gamma} = \frac{\sigma_s}{\varepsilon_s}$$

Poisson's ratio (ν)

The Poisson's ratio (ν) is the ratio of the lateral to axial strains. Theoretically, isotropic materials (identical in all directions) will have a value for Poisson's ratio of 0.25. The maximum value of n is 0.5, which denotes no volume change during deformation. It is also used to relate shear and elastic modulus using the equation below:

$$E = 2G(1 + \nu)$$

Transversely isotropic coefficient of thermal expansion (CTE) data

The transversely isotropic coefficient of thermal expansion (CTE) specifies thermal expansion properties of the material. This data combined with the mechanical data is used to account for variation in properties parallel and perpendicular to the direction of flow.

Shrinkage Properties

Material shrinkage properties are important to be taken into account for the accurate prediction of the parts' final dimensions. Moldflow Plastic Insight uses the following shrinkage models depending on the type of mesh used and if shrinkage data is available or not.

Shrinkage models used in Midplane and Fusion

There are several options of shrinkage models that can be used during the simulation. The best model is set by default for material tested.

Residual strain shrinkage prediction method

Historically, the residual strain method is the older of the two shrinkage prediction methods employed by Moldflow in its warpage analysis product for Midplane and Fusion. In this method we calculate shrinkage strain left in the parts after they have been molded and it requires shrinkage data because the coefficients of the equations have components which relates directly to the parts's final dimensions.

Residual stress shrinkage prediction method

In this method, rather than calculate shrinkage strain we directly calculate a residual stress distribution for each element. This distribution provides the stress across the thickness of each element in directions parallel and perpendicular to flow. This stress distribution is then input to the stress analysis program to obtain the deflected shape of the part.

a. Uncorrected - doesn't require shrinkage data.

b. Corrected (**C**orrelated **R**esidual **I**n **M**old **S**tress) - requires shrinkage data.

The CRIMS method, in general provides considerably more accurate prediction of shrinkages and hence part deflections can be obtained than using the residual strain method.

Observed nominal shrinkage

The observed nominal shrinkage shows the average shrinkage value, parallel and perpendicular to the direction of flow.

Observed shrinkage

The observed shrinkage shows the maximum and minimum shrinkage values, parallel and perpendicular to the direction of flow.

Shrinkage prediction method used in 3D

Warpage is caused by variations in shrinkage throughout the part. Two types of shrinkage variations are considered: shrinkage variations from region to region and shrinkage variation in different directions.

A hybrid element scheme has been designed for the 3D warpage analysis. 4-node first-order tetrahedral elements are used in the 3D chunky areas, and 10-node second-order tetrahedral elements are used in the thin-walled areas. Transitional 5-9 node tetrahedral elements are used in the transitional areas which connect the thin-walled and chunky areas.

For fiber-filled polymers, the flow-induced fiber orientation is also simulated. Then the moduli, Poisson's ratios and thermal expansion coefficients of the composite material can be determined using the calculated fiber orientation and mechanical properties of polymer matrix and the fiber. This information is put into structural tetrahedral elements, in which the 3D orthotropic stress-strain relationship is used.

3D warpage simulation normally requires significant computational time particularly if the number of elements is very large and if there is a big thin-walled region. An efficient preconditioned conjugate gradient iterative solver is implemented for reducing the memory requirement and computational time.

Thermoplastic material families & abbreviations

Below are the thermoplastic material families available in Moldflow programs as well as the resin's abbreviation inside each family.

Table 34: Thermoplastic material families & abbreviations

Family Name	Family abbreviation
Acetals (POM)	POM, POM-HI
Acrylics (PMMA)	PMMA, PMMA+EA
Acrylonitrile copolymers (ABS, ASA, ...)	ABS, AES, AMA, ASA, MABS

Table 34: Thermoplastic material families & abbreviations

Family Name	Family abbreviation
Blends (PC+PBT, PC+ABS, ...)	ABS+PA, ABS+PA6, ABS+PA66, ABS+PBT, ABS+PC, ABS+PVC, ASA+PC, EPDM+PP, PA+ABS, PA+PO, PA+PP, PA+PPE, PA6+PA66, PA6+SAN, PA66+PPE, PA6I/6T, PA6T/6I, PBT+ASA, PBT+PC, PBT+PET, PC+ABS, PC+AS, PC+ASA, PC+PBT, PC+PET, PC+PS, PC+PSU, PC+SAN, PEI+PC, PET+PBT, PET+PC, PMI+PMMA, PMMA+PC, PP BLEND, PP+EMPP, PP+EPDM, PP+EPR, PP+ERDM, PP+PA6, PP+PA66, PP+PE, PP+PMMA, PP+PPE, PP+PS, PP+SEBS, PP+TPE, PP+TPO, PPE+HIPS, PPE+PA, PPE+PA6, PPE+PP, PPE+PS, PPE+SB, PPO+PA, PPO+PS, PS+PMAA, PS+PMMA, PVC+ABS, SMA+PPE, TPU+PC
Cellulosics (CA, CAP, CN, ...)	CAP
Copolyesters (PCTG)	PCTG
Copolyester (PETG)	PETG
Ethylene copolymers (EP, EPDM, ...)	E/VA, EMPP, EVA, EVOH
Fluoropolymers (Teflon, …)	PFA
Ketone-based resins (PEEK, PAEK, ...)	PEEK, PK
LCP	LCP
Polyamides (NYLONS, PPA, ...)	PA, PA11, PA12, PA12/MACMI, PA12/ MACMI+PA12, PA12/X, PA46, PA6, PA6+PA6I/6T, PA6/6, PA6/6T, PA6/X-HI, PA610, PA612, PA63T, PA66, PA66+PA6, PA66+PA6I/6T, PA66+PA6T/6I, PA66/6, PA66/6I, PA66/6T, PA666, PA6T, PA6T/66, PA9T, PAI, PAMACM12, PAMACM12+PA12, PAMACMI/12, PAMACMI/12+PAMACM12, PPA
Polycarbonates (PC)	PC
Polyesters (PET, PBT, …)	PBT, PBTP, PCT, PCTA, PET
Polyethylenes (PE)	HDPE, LDPE, LLDPE, LMDPE, MDPE, PE, UHMWPE, VHMWPE
Polyimides (PI, PEI, ...)	PEI, TPI
Polyphenylenes (PPE, PPS, PPO, ...)	PPE, PPE+PA66, PPO, PPS
Polypropylenes (PP)	HCPP, PP, PP(CO), PP(HOMO), PP(ICP), PP(RCP)
Polyurethanes (PUR)	PUR, PUR (PU)
Styrenics (PS, SAN, SBR, …)	GPPS, HIPS, IR+HIPS, IR-HIPS, PC+HIPS, PS, PS-SY, SAN, SB, SEBS, SMA, SPS, SPS+PA, SVA
Sulfone-based resins (PSU, PES, …)	PES, PPSU, PSU, PSU+PPSU

Table 34: Thermoplastic material families & abbreviations

Family Name	Family abbreviation
Thermoplastic elastomers (TPO, TPU, TPR, …)	POE, RTPU, SEBS+TPE, TEEE, TEO, TPE, TPE-S,TPEE, TPO, TPR, TPU
Thermoplastic vulcanizates (TPV, …)	TPV
Vinyl-based resins (PVC, PVAC, PVAL, ...)	PPVC, PVC, PVC-F, PVDF
Miscellaneous	ACS, ALLOY, AP, APO, BUTYRATE, COC, COP, E/BA, EMAA, EPDM, ESTER, IONOMER, MIM, MIPS, MISC, MPPO, PAR, PAT, PB, PEEL, PEN, PI, PLA, PMP, POLYOLEFIN, PPV, PU, RPVC, TPX, TVO, TX, UPVC, WAX

What You've Learned

After completing this unit you have learn that plastics are in reality thermoplastics and that polymers can be thermoplastics or thermosets.

Thermoplastics can be divided even more into amorphous and semi-crystalline polymers based on their morphology. Simulation software takes into account the morphology of the polymers thru the usage of the specific volume data (also know as pvT diagram).

For the accuracy of the simulation results the following material data is required:

- Processing conditions: mold temperature, melt temperature, ejection temperature, maximum shear stress and maximum shear rate.
- Rheological properties: viscosity, juncture loss method coefficients, transition temperature and melt flow rate.
- Thermal properties: specific heat and thermal conductivity.
- Physical properties: melt density, solid density and specific volume (pvT diagram).
- Mechanical properties: elastic modulus, poisson's ratio, shear modulus and transversely isotropic coefficient of thermal expansion data.
- Shrinkage properties: shrinkage model, observed nominal shrinkage and observed shrinkage.

Index

C

D

E

F

T