Autodesk® Moldflow® Insight 2012

AMI Modeling the cooling system

Revision 1, 22 March 2012.

This document contains Autodesk and third-party software license agreements/notices and/or additional terms and conditions for licensed third-party software components included within the product. These notices and/or additional terms and conditions are made a part of and incorporated by reference into the Autodesk Software License Agreement and/or the About included as part of the Help function within the software.

Contents

Chapter 1 **Cooling system** . . . 1

Mold Surface Wizard . . . 2

Mold Surface Wizard . . . 2

Mold Surface Wizard . . . 2

Chapter 2 **Import and export** . . . 3

Import cooling systems . . . 3

Import cooling systems . . . 3

Export cooling systems . . . 4

Export cooling systems . . . 4

Chapter 3 **Cooling considerations** . . . 5

Cooling time . . . 5

Ejection temperature . . . 6

Heat transfer . . . 6

Uniform cooling . . . 7

Effective heat extraction . . . 9

Heat transfer due to coolant flow. 9
Heat transfer effectiveness. 10
    Heat transfer effectiveness 11
Cooling system equations. 12
Interface conductance. 12
    Interface conductance. 14
Mold materials. 14

Chapter 4 **Cooling circuits**. 16
Cooling circuits. 17
    Creating cooling channels manually. 17
    Meshing manually created cooling channels. 18
Cooling Circuit Wizard. 18
    Cooling Circuit Wizard. 18
    Cooling Circuit Wizard. 19
Cooling circuit design. 20
Cooling circuit parameters. 21
    Cooling circuit parameters. 23
Cooling planes. 23
    Cooling planes. 24
Parallel and series cooling circuits. 25
Cooling circuit design references. 27

Chapter 5 **Cooling components**. 28
Hoses. 28
    Hoses. 28
Bubblers. 29
    Bubblers. 30
    Bubblers. 33
Baffles. 33
    Baffles. 34
    Baffles. 36
Flow control valves. 36

Chapter 6 **Coolant**. 37

Common coolants. 37
Common coolants. 38
Coolant flow. 38
Typical coolant temperatures. 40
Determining the coolant temperature. 40
Coolant inlets. 41
Coolant inlets. 41
Coolant inlets. 41
Coolant inlet temperature. 42
Coolant inlet temperatures. 42
Pump capacity. 43
Use of tower water. 43
Fouling of water lines. 44

Cooling system

1

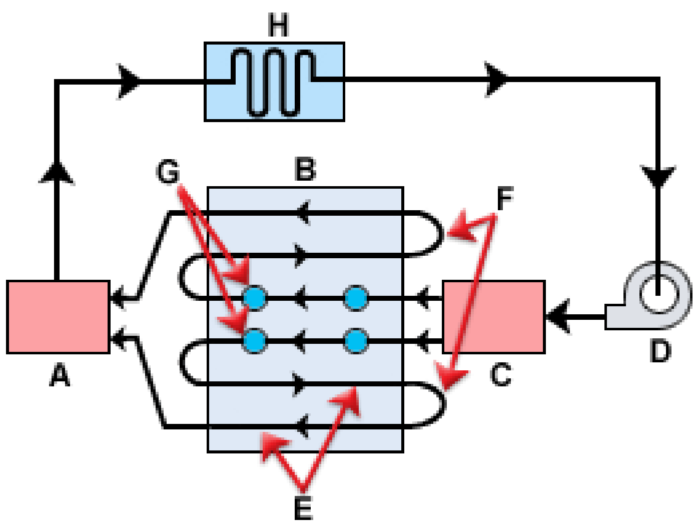
The cooling circuit is comprised of the cooling channels, various components added to the cooling channels to facilitate cooling, and the coolant which flows through the system.

The overall requirements for cooling the part will always be a compromise between uniform cooling to assure part quality, and fast cooling to minimize production costs. The degree to which consideration outweighs the other is dependant on the functional requirements of the part.

To channel coolant into high heat-load areas, it may be necessary to design intricate cooling systems.

Cooling system components

The following diagram of a cooling system illustrates typical cooling elements.



A = Collection manifold
B = Mold
C = Supply manifold
D = Pump
E = Cooling Channels
F = Hoses
G = Baffles

H = Temperature controller

Mold Surface Wizard

The **Mold Surface Wizard** is a tool for quickly modeling a cuboid mold outer surface around the existing model for the purposes of improving the accuracy of a Cool analysis.

The **Mold Surface Wizard** determines the dimensions of the required mold surface from the supplied inputs, creates a region for each face of the cuboid, assigns the property Mold block surface to the regions, and then meshes the regions. The mold surface regions and elements are assigned to separate layers so you can control their visibility and properties.

Mold Surface Wizard

In a Cool analysis, the flow of heat from the mold is affected by the exterior boundary of the mold.

Simulating the mold block heat flow

To simulate the effect of heat flow from the exterior boundary of the mold, you need to model the shape of the exterior of the mold as a mold block surface.

Enter the mold's dimensions in the Mold Surface Wizard (**Geometry tab > Create panel > Mold Surface Wizard**) to quickly create a surface of the appropriate size.

Mold Surface Wizard

The Mold Surface Wizard is a tool for helping you to model a cuboid surface around the outside of your model. The mold block surface dialog allows you to edit the properties of the mold block.

Mold Surface Wizard

The **Mold Surface Wizard** is a tool for quickly modeling a cuboid mold outer surface around the existing model for the purposes of improving the accuracy of a Cool analysis.

To access this wizard, click **Geometry tab > Create panel > Mold Surface**.

The **Mold Surface Wizard** determines the dimensions of the required mold surface from the supplied inputs, creates a region for each face of the cuboid, assigns the property Mold block surface to the regions, and then meshes the regions. The mold surface regions and elements are assigned to separate layers so you can control their visibility and properties.

Import and export

2

You can save time and effort by importing cooling circuits from other studies to use in your current project. Cooling circuits created in your current project can also be exported for reuse in other studies.

Import cooling systems

Different file formats will import different cooling attributes.

- When importing a cooling system from an *.adv file, you must first import the *.adv file, and then export it as a *.sdy or *.udm file. The cooling system will then be available for import.
- When importing a *.sdy file, the process will import only the cooling elements in the study file. These include all channels, hoses, baffles and bubblers.
- When importing a *.udm file, the process will import only the cooling elements in the .udm file. These include all channels, hoses, baffles and bubblers.
- When importing an *.igs or *.iges file, the process will import each segment of the cooling system as a centre or axis line, and apply default cooling channel dimensions. Baffles and bubblers are converted into cooling channels, and these elements will need to either be defined or added manually after import.

Import cooling systems

Cooling systems modeled in a CAD system can be imported and used within the Moldflow product.

Import cooling systems from a CAD system

Autodesk Moldflow Insight allows you to import a cooling system, and associated runner system, from different file formats.

NOTE: In many cases the cooling system will be imported as a centre (axis) line and the elements will need to be defined after import.

---

1. Click then click **Open > Import**.
   The **Import** dialog appears.

2. Navigate to the location where you saved your model, including the cooling system.
3. Select the appropriate file type from the drop-down box, select your file, then click **Open**.
   The **Import** dialog appears.
4. Select the mesh type, then click **OK**.

Export cooling systems

Different file formats will export different cooling attributes.

- When exporting an **.igs** or **.iges** file, each segment in the cooling system is exported as a cooling channel. Elements like hoses, baffles or bubblers will not export. Instead, these elements will convert to cooling channels.
- When exporting a **.udm** file, elements like channels, hoses, baffles and bubblers will export as part of the cooling system.
- When exporting your cooling system, it is saved as an ASCII IGES (*.igs), or ASCII UDM (*.udm) file.

Export cooling systems

The cooling system design used in an analysis can be exported as an IGES file.

Export cooling systems as IGES

Autodesk Moldflow Insight allows you to export your cooling system, and associated runner system, into a CAD package as an IGES file.

NOTE: Although a cooling system is made of curves, nodes and beam elements, only curves will be exported. If your model contains iges surfaces, they will also be exported.

---

1. Click then click **Export > Study & Results**.
   The **Export** dialog appears.
2. Navigate to the location where you want the export file to be saved.
3. Select **ASCII IGES (*.igs)** in the **Save as type** drop-down box, then click **Save**.
   A warning dialog appears.
4. Click **Yes**.

Your cooling system has been exported and can be read from a CAD package or from Autodesk Moldflow Insight.

Cooling considerations 3

It is important that when designing plastic parts you consider the various aspects of cooling, and how they affect the finished product.

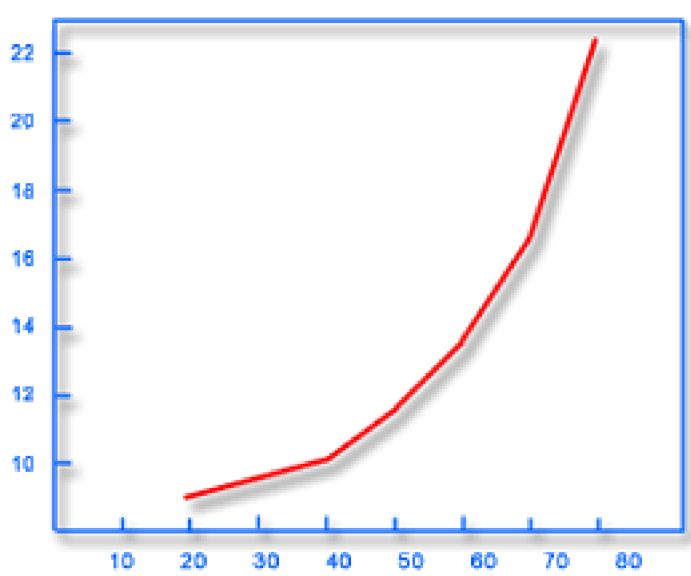
Cooling time

Cooling time is the time after the end of packing until ejection. Usually, the material at the center section of a part wall reaches its freeze temperature (Vicat softening point) and becomes solid during cooling time. Cooling time usually represents 80 percent of the total cycle time.

Two major factors that affect the cooling time are melt temperature and mold temperature. Both may need to be optimized to obtain a high quality part. Increasing either the melt or mold temperature increases the cooling time because it takes longer for the frozen layer to reach the required thickness.

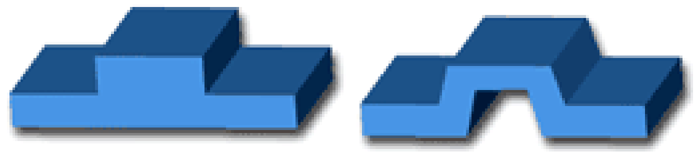
Lower mold temperatures = shorter cycle time

Lower mold temperatures result in shorter cycle times, which leads to higher productivity. The following diagram illustrates how increasing the mold temperature increases the cycle time.



Part thickness

Cooling time increases rapidly with wall thickness, so avoid thick part walls to maintain an economically acceptable cooling time. Part thickness should be as uniform as possible. In the following diagram, the part on the left has a thick wall section. This part will take longer to cool than the part on the right



NOTE: Cooling system equations have additional information about cooling times and part thicknesses.

Ejection temperature

The ejection temperature is the temperature below which the plastic is solid.

When the whole of the part has reached the ejection temperature, the part can be ejected from the mold with no adverse effect on its quality.

Since the cooling time frequently comprises some 80 percent of the cycle time, any reduction in cooling time will significantly reduce the cycle time. Frequently, the cycle time is determined by the time taken to reduce the part temperature to a level at which it can be safely ejected without compromising the quality of the part. The temperature at which a part can be ejected from the mold is affected by a number of factors.

Successful ejection requires the part to be stiff enough to resist any tendency to warp caused by shrinkage variations and residual stresses, and stiff enough to resist the local forces on the part from the ejection system. The geometry of the part, the mold finish, and the degree to which the cavity has been packed during the filling and packing phases can all affect the cycle time.

The overall requirements for cooling the part will always be a compromise between uniform cooling to assure part quality, and fast cooling to minimize production costs.

Heat transfer

The mold can be thought of as a heat exchanger because heat passes into the mold and transfers out of it.

The mold's primary energy input is hot plastic injected into the cavity. Hot runners can also be a an energy source.

During mold filling, heat is lost by the following three mechanisms:

Coolant flow in the cooling circuits	About 80-95 percent of the heat introduced into the mold by the plastic melt is transferred by conduction through the metal to the surface of the cooling channels and dissipated into the heat transfer fluid.
Conduction to the injection molding machine mold	Most of the heat will be extracted by coolant flow. When running the mold hot, the heat lost through the mold and to the atmosphere may exceed heat input from the plastic melt.
Convection and radiation to the atmosphere	Convection on the mold surfaces and conduction into the molding machine are of only minor importance, accounting for typically 5-15 percent of the total heat flow. The radiative heat transfer should be considered only when the temperature of the mold is high, that is, greater than 85°C, because the radiative heat flow is typically less than 5 percent of the total amount.

In addition to the heat input from the plastic melt, the hot runners and manifolds can also contribute heat to the mold. Cases where the coolant is at a temperature well above the ambient temperature also contribute heat.

Air gaps

A layer of air can impair the effective transfer of heat. Therefore, take steps to eliminate any air gaps between the mold insert and molding plates, and any air pockets in the cooling channels.

Temperature difference

The temperature difference on opposite sides of the part should be kept to a minimum and should not exceed 10°C for parts that require a tight tolerance.

Uniform cooling

Uniform cooling helps you achieve effective heat transer .

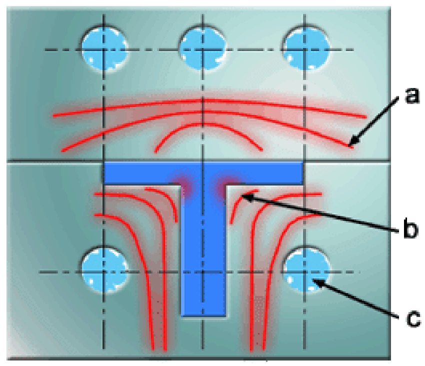
Effective heat transfer is achieved when the following conditions are met:

- The coolant flow is turbulent.
- The area of the cooling surface is great enough.
- The temperature difference across the coolant/metal interface is 2–5°C.
- The inlet/outlet temperature of the coolant varies between 2–3°C.

When these conditions apply, the temperatures on the mold surface are determined by heat transfer through the metal to the coolant/metal interface. This is a function of the conductivity of the metal and the geometry of the mold.

Surface temperatures are controlled by the position of the cooling circuits relative to the surface of the mold.

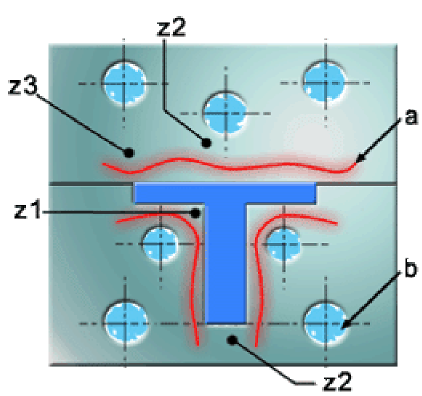
In the following diagram, the surface temperature is highest at the internal corners and lowest on the thin top bar. To achieve a uniform temperature, the cooling channels must be placed as near as possible to hot spots and away from surfaces that have a tendency to run cold.



where:

- **a** - Isotherms
- **b** - Hot spot
- **c** - Cooling circuit

In the following diagram, an extra cooling channel has been placed as close as possible to the surface of mold. The hot spot in zone 1 has been eliminated. The adjustment of cooling channels around zone 2 accounts for the heat load which comes from the thick leg section of the part.



where:

- **a** - Isotherms
- **b** - Cooling circuit
- **z1**, **z2** and **z3** - Zone 1, zone 2, and zone 3

Effective heat extraction

Effective heat extraction can reduce the cycle time.

Effective heat extraction requires the consideration of the following variables.

Inlet/outlet temperature

The coolant flow rate should be high enough to ensure that the temperature at the outlet is within 2-3°C of the inlet temperature.

Turbulence

Effective heat extraction requires the fllow of coolant flow to be turbulent. Turbulence is indicated by the Reynolds number, which is calculated from the diameter of the cooling channels, the flow rate, and viscosity of the coolant. A Reynolds number of 10,000 is recommended.

Surface area

The surface area of the cooling channel must be great enough to ensure that the temperature rise across the channel is 2-5°C.

Increasing the length or number of the channels improves the area available for heat transfer. This results in a higher pressure drop in the channel. If the diameter of the cooling channel is increased, a higher flow rate is required to achieve turbulence.

A balance has to be struck between the diameter and length of the cooling channels, and the pressure and volume characteristics of the cooling pump.

When these conditions have been optimized, the temperature rise across the metal is controlled by the placement of cooling circuits. Although, ideally, this should be no more than 5°C, it is more realistic to expect a temperature rise across the metal of 10-30°C.

Heat transfer due to coolant flow

The effect of heat transfer increases as the flow of coolant changes from laminar flow to turbulent flow.

For laminar flow, heat can be transferred only by means of heat conduction from layer to layer. However, in turbulent flow, the mass transfer in the radial direction enables the heat to be transferred by both conduction and convection. As a result, the efficiency increases dramatically.

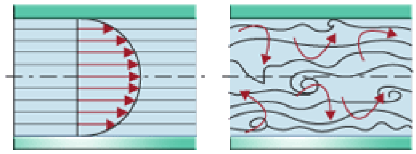


Figure 1: Left: Laminar flow, Right: Turbulent flow

Since the increase of heat transfer will diminish as the coolant flow becomes turbulent, there is no need to increase the coolant flow rate when the *Coolant flow* on page 38 exceeds 10,000. Otherwise, the small, marginal improvement in heat transfer will be offset by the higher pressure drop across the cooling channels, along with more pumping expense.

Once the flow becomes turbulent, a higher coolant flow rate brings diminishing returns in improving the heat flow rate or cooling time, while the pressure drop and pumping expenses are drastically increasing. This concept is shown below.

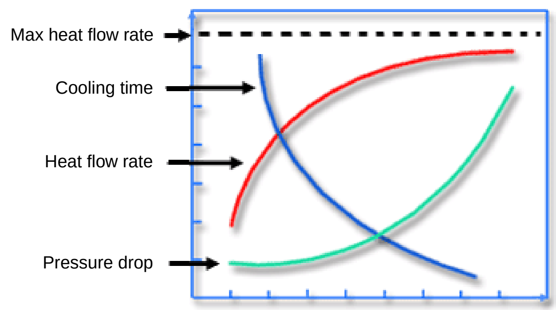


Figure 2: Flow rate

NOTE: It is important to make sure that the coolant reaches turbulent flow everywhere in the cooling system. Autodesk Moldflow analyses can help you identify and correct problems such as stagnated cooling channels, by-passed cooling channels, and high pressure drops in some cooling circuits.

Restrictive flow plugs

Coolant will take the path of least resistance to flow. Use a restrictive flow plug in certain cooling channels to redirect the flow of coolant to other cooling channels that have a high heat load.

Heat transfer effectiveness

A heat transfer effectiveness (HTE) value can be assigned to mold surfaces to indicate the effectiveness of heat transfer through specific mold surfaces.

This value is used to specify the effectiveness of heat transfer for baffles, bubblers, cooling circuits (channels) or connectors for a Cool analysis.

HTE values are automatically set for appropriate cross sections, such as channels, baffles, bubblers and hoses. Rectangular cross sections may need to be set manually.

NOTE: HTE must be assigned to any bubblers and baffles that exist in your design, this is done automatically. Bubblers require a HTE value of 1 on the outside surface and a value of 0 on the inside surface, and semi-circular baffles should be assigned an HTE value of 0.5.

HTE Values

The program will set a default HTE value of 1, indicating the maximum heat transfer. You can also set the HTE value to 0, indicating that there is no heat transfer through these surfaces. A value of 0.5 is assigned when only half of the surface parameter is used to transfer heat (for example, baffles).

When all the nodes connected to a surface are defined to be at the same temperature, the HTE value given to the surface is irrelevant because there can be no heat flow through the nodes. Thus the default value of HTE need not be changed.

The HTE value of 1, which is assigned to cooling circuit surfaces by default, indicates that the mold feature only absorbs heat. If an HTE of 0 is assigned to a feature, it indicates that the feature absorbs no heat from the mold.

Heat transfer effectiveness

Assign a heat transfer effectiveness (HTE) value to mold surfaces to indicate the effectiveness of heat transfer through specific mold surfaces.

Editing heat transfer effectiveness properties

The heat transfer effectiveness (HTE) is a required input for a Cool analysis that specifies the heat extraction efficiency of mold surfaces, including cooling circuits (channels), baffles, bubblers, or connectors. An HTE value of 1 indicates that the mold feature only absorbs heat, while an HTE value of 0 indicates that the feature absorbs no heat from the mold.

1. Click .
2. Click on the mold feature that you want to assign an HTE for.
3. Right-click on the highlighted mold feature and select **Properties**.

   If the mold feature is not yet created, click **Geometry tab > Properties panel > Assign** and create a **New** mold feature.
4. In the **Heat Transfer Effectiveness** box, enter the required value.
5. Click **OK**.

Cooling system equations

Theoretically, cooling time is proportional to the square of the heaviest part wall thickness and the largest runner diameter raised to the power of 1.6, and inversely proportional to the thermal diffusivity of the polymer melt.

Cooling time

The thermal diffusivity of polymer melt is defined as: R 2 p ⊠c v where:

- R2 is the thermal conductivity
- p is the density
- cv is the specific heat constant volume

In other words, doubling the wall thickness quadruples the cooling time.

Reynolds number and coolant flow

The type of coolant flow can be determined by the Reynolds number (Re), as listed in the following table.

Reynolds number (Re)	Type of flow
10,000 < Re	Turbulent
2,300 < Re < 10,000	Transition
100 < Re < 2,300	Laminar
Re < 100	Stagnated

The Reynolds number (Re) is defined as: Re=p⊠U⊠⊠where:

- p is the density of the coolant
- U is the averaged velocity of the coolant
- d is the diameter of the cooling channel
- ⊠s the dynamic viscosity of the coolant

Interface conductance

Interface conductance is a measure of the rate of heat transfer across the surfaces represented by inserts and parting planes.

The higher the rate of heat flow for a given temperature and area, the larger the interface conductance. If there is ineffective heat transfer due to air gaps, for example, the interface conductance value is very small.

How interface conductance is used

Interface conductance is an optional attribute that you can assign to a mold insert or parting plane for a Cool analysis. Where insert surfaces have low values of interface conductance to represent assembly clearances, the temperature difference shows the difference in temperature from one side of the gap to the other. Where inserts directly contact the plastic, the insert temperatures are the same as the plastic.

When to specify interface conductance

The interface conductance value is determined primarily by the finish (smoothness) of the interface and the fit of the two metal surfaces. For inserts that have a good contact with the main mold material, it is not necessary to assign an interface conductance because the analysis program assumes a high value. Furthermore, insert surfaces directly in contact with plastic surfaces are assumed to have a very high value of interface conductance and any values that you assign to such surfaces will be ignored.

Typical interface conductance values

Contact Type	Contact Pressure [MPa]	Surface Finish [μm]	Thermal Conductance [W/m^{2}C] +/- 10%
Steel/Air/Steel (no gap)	0	Normal grinding	2500
Steel/Air/Steel (no gap)	7	Normal grinding	21000
Steel/Air/Steel (no gap)	15	Normal grinding	38000
Steel/Air/Steel (0.5 mm gap)	N/A	N/A	50
Steel/Air/Steel (1.0 mm gap)	N/A	N/A	25
Steel/Grease/Steel (Ejector pin or sliding core)	0	Lapped 2μm radial gap	75000
Steel/Grease/Steel (Ejector pin or sliding core)	0	Lapped 5μm radial gap	30000
BeCu/Air/Steel (T/C = 130 W/mC)	330	Normal grinding	3700000

Interface conductance

Interface conductance is a measure of the rate of heat transfer across the surfaces represented by inserts and parting planes.

Editing interface conductance properties

The interface conductance value quantifies the ability of heat to pass between the insert, or parting plane, and the surrounding mold material. Interface conductance is an optional input for a Cool analysis that can be assigned to a mold insert or parting plane.

1 Click .
2 Click on the mold insert or parting surface that you want to assign an interface conductance for.
3 Right-click on the highlighted insert or parting surface and select **Properties**.

   If the insert or parting surface isn't created yet, click **Geometry tab > Properties panel > Assign** and create it.
4 In the **Interface Conductance** box, enter the required value.

   You can also set the insert material in this dialog.
5 Click **OK** to close the dialog and save your changes.

Mold materials

To run a Cool analysis, all mold model surfaces must be defined with a mold material. This allows an accurate estimate of cooling rates in the design, using the properties associated with that mold material.

Effects on cooling performance

Cooling system performance is affected by how quickly heat moves from the plastic part to the mold cavity surface. This is affected by the material properties, the difference between melt and mold surface temperature, and the quality of the contact between the cooling plastic and the mold material.

Cooling system performance is also affected by the transfer of heat through the mold material to the coolant channels. Heat transfer is affected by turbulence in the coolant as it flows through the mold material, by the coolant inlet temperature, the coolant properties and the coolant flow rate.

Thermal conductivity

The thermal conductivity of a mold material is also important to the performance of a cooling system. The units for thermal conductivity are; watts per meter °Kelvin [W/mK] in the metric system, or, Btu per hour foot

°Fahrenheit [(Btu/hr)/ft/F)] in US units. High values of thermal conductivity indicate that the material is a good conductor.

Some typical thermal conductivities for pre-defined mold materials are shown below:

Mold Material	Conductivity W/m C	Specific Heat J/KgC	Density kg/m3
Stainless Steel	24	460	7800
P20 Steel	29	460	7800
Low TC Copper Alloy	90	420	8400
Medium TC Copper Alloy	160	420	8400
Aluminum	170	780	2800
High TC Copper Alloy	250	420	8400

NOTE: A Fill + Pack analysis incorporates a temperature boundary condition on the mold wall and so returns the same result regardless of the mold material specified. To investigate the effect of altering the mold material, a Cool analysis must be run.

Cooling circuits

4

Cooling circuits are used in a Cool analysis to deliver coolant to areas of the mold that would not otherwise cool effectively.

A cooling circuit consists of a set of connected two-point beam surfaces which make up the cooling system.

The placement of cooling channels is restricted by mechanical constraints such as ejector pins and metal inserts. Information from Cool analyses can be used to evaluate each design. When you are designing a cooling system, consider the coolant inlet, the circuit type, and the circuit location.

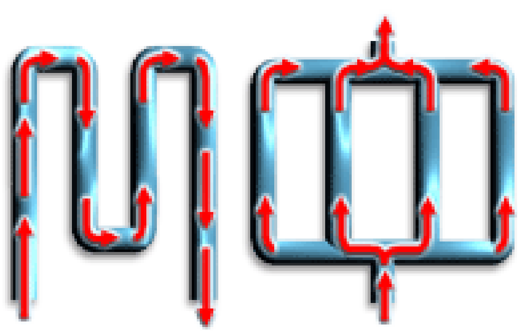
Coolant inlet

Prior to running a Cool analysis, you need to identify and set all cooling circuit inlets.

Circuit type

A series circuit that can achieve even coolant flow rate and heat transfer is usually preferable to a parallel circuit.

If it is necessary to use a parallel circuit, each branch should be balanced for local heat load. Poorly designed parallel circuits can have branches where there is little or no flow. The flow in each branch should be controlled so that all of the cooling circuit coolant flowing through them, and that the flow is turbulent for maximum cooling efficency. The following diagram illustrates a series cooling circuit on the left and a parallel circuit on the right.



Circuit location

In general, locate cooling circuits at a distance of about 2.5 times their diameter from the plastic. This will give fairly uniform cooling over the part. In some cases, however, it may be necessary to locate the line closer or further from the part, depending on how much heat is to be removed. Cooling circuits should be close in areas that concentrate heat, such as internal corners and ribs. Cooling circuits can be positioned further away in areas that have less heat content, such as thin sections.

Cooling efficiency

Often molds contain ribs and cores that are very difficult to cool. Placing bubblers, baffles, or metals of high conductivity in these areas greatly improves conduction through the core to the cooling channel.

Do not use small inlet channels to feed larger channels. Remember that the only channels in which turbulent flow is required are the circuits that are actually cooling the part. If a small inlet line feeds a large cooling circuit, achieving turbulence in the large circuit must be accompanied by a large pressure drop over the smaller line. This is a waste of pumping power.

Cooling elements must be assigned a value of heat transfer effectiveness to represent their ability to accept heat from the mold. For most situations a default value is applicable.

Cooling circuits

Cooling circuits can be added and deleted manually.

Creating cooling channels manually

Cooling channels can be created either manually or using the Cooling Circuit Wizard

To model a cooling circuit manually, you must create the beams which make up your cooling circuit, assign their properties (different sections of the circuit may be different diameters), mesh the channels and assign coolant inlets.

1. Create the curves which describe the desired cooling circuit layout.
2. Select the desired curve(s).
3. Click **Geometry tab > Properties panel > Assign**.

   The **Assign Property** dialog appears.
4. Click **New** and select **Channel** from the drop-down list.

   The **Channel** dialog appears.
5. Select the properties as needed.
6. Click **OK** twice to close all dialogs.
7. Continue to assign properties to all channels.

Sections which extend outside the mold may be assigned as **Hose** (heat transfer effectiveness=0).

NOTE: The cooling channels must now be meshed. For Cool (FEM), this should be done when you mesh the mold.

Meshing manually created cooling channels

Once you have created cooling channels, manually, they must be meshed.

1 Click **(Mesh tab > Mesh panel > Generate Mesh)**.

The **Generate Mesh** dialog appears. Notice the **Global edge length** is already defined. You can change this, and preview its effect.

TIP: For better results, modify the Global edge length to obtain a length/diameter ratio of 2.5 to 3. For example, the Global edge length should be between 25–30 mm for a channel that has a diameter of 10 mm.

---

2 Click the **Advanced** button.

3 Click **Mesh Now**.

The default color for a cooling channel is blue.

4 Assign coolant inlets to the ends of the channels where necessary.

Cooling Circuit Wizard

The **Cooling Circuit Wizard** helps you create a pair of basic channels and hoses, to produce a simple cooling system quickly.

The **Cooling Circuit Wizard** can help you create a cooling system quickly and easily. Simply follow the prompts as the Wizard takes you through the steps. Once you are comfortable with the process you can create your own cooling system manually, if you prefer.

Cooling Circuit Wizard

The Cooling Circuit Wizard assists in the design of the cooling circuit.

Creating cooling circuits with the Cooling Circuit Wizard

The **Cooling Circuit Wizard** can be used to create a pair of serial cooling circuits above and below your part.

Your part must be oriented in the XY direction to use the Wizard.

1 Click **Geometry tab > Create panel > Cooling Circuit**.

The **Layout** page of the Cooling Circuit Wizard appears. This page is used to specify the layout of the cooling circuits.

TIP: Click **Use Defaults** to apply default specifications to both pages of the Wizard.

---

2. Enter a value into the **Please specify the channel diameter to use:** text box.
3. Enter a value into the **How far above and below the part would you like the circuits to be created?** text box.

   This is measured from the part surface to the closest point on the cooling channel (not the midpoint of the cooling channel).
4. Select **X** or **Y** to choose how you would like the circuits aligned with your part.
5. Click **Next**.

   The **Channels** page of the Cooling Circuit Wizard appears on screen.
6. Enter a value into the **Number of channels** text box.
7. Enter a value into the **Distance between channel centers** text box.
8. Enter a value into the **Distance to extend beyond part** text box.
9. Optional: Click **Preview** to preview the cooling channels layout.
10. Optional: Select the **Delete existing circuits first** checkbox if you would like to delete existing circuits before creating a new circuit.
11. Optional: Select the **Connect channels with hoses** checkbox if you would like the sections which extend beyond the part to be assigned as hoses (heat transfer effectiveness = 0).
12. Click **Finish** to create the channels and close the wizard.

    The newly created cooling system will appear on the model.

Cooling Circuit Wizard

These dialogs are used to create a basic cooling system so you can test your design.

To access this dialog, click **Geometry tab > Create panel > Cooling Circuit**.

Cooling Circuit Wizard—Layout

The **Cooling Circuit Wizard** allows you to quickly and easily create an initial cooling system design suitable for Cool analysis. This initial design can be enhanced using the modeling tools provided in the **Geometry** tab. To access this wizard, click **Geometry tab > Create panel > Cooling Circuit**.

The **Layout** page of the Cooling Circuit Wizard is used to input information about the general layout of the cooling channels in relation to the part.

Cooling Circuit Wizard—Channels

The **Cooling Circuit Wizard** allows you to quickly and easily create an initial cooling system design suitable for Cool analysis. This initial design can be enhanced using the modeling tools provided in the **Geometry** tab. To access this wizard, click **Geometry tab > Create panel > Cooling Circuit**.

The **Channels** page of the Cooling Circuit Wizard is used to input specific information about the cooling channels in the cooling system design.

Cooling circuit design

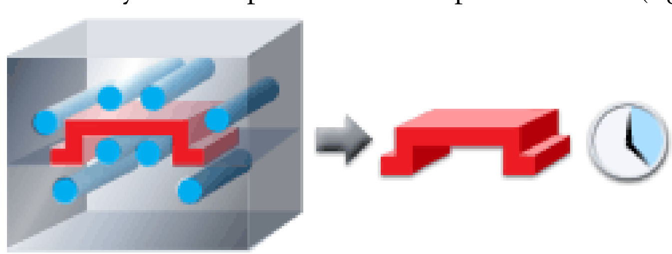
Mold cooling accounts for more than two-thirds of the total cycle time in the production of injection molded thermoplastic parts. An efficient cooling circuit design reduces the cooling time, which in turn increases overall productivity. A well designed circuit achieves uniform cooling, improving part quality by reducing residual stresses and maintaining dimensional accuracy and stability.

The primary factor governing production costs is cycle time, and the cycle time is governed by the material's ejection temperature.

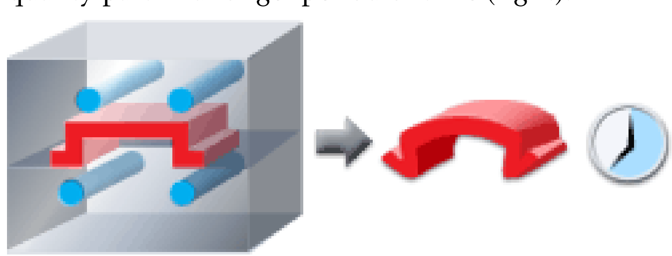
To ensure part quality, consider the following factors:

- Surface finish
- Residual stresses
- Crystallinity
- Thermal bending

The following diagram shows how an effectively cooled part (left) leads to a correctly molded part in a shorter period of time (right).

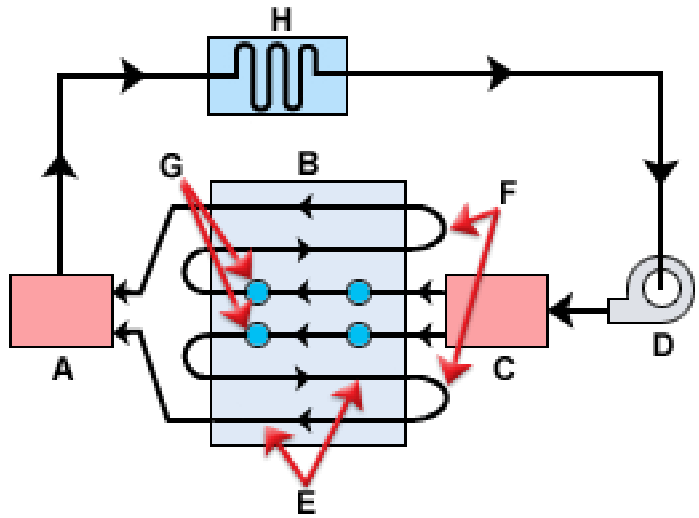


The following diagram shows how a poorly cooled part (left) leads to a low quality part in a longer period of time (right).



Cooling system components

A cooling system typically consists of the following items:



A - Collection manifold
B - Mold
C - Supply manifold
D - Pump
E - Cooling channels
F - Hoses
G - Baffles
H - Temperature controller

Cooling circuit parameters

The flow rate of the coolant is an important aspect to consider when preparing a Cool analysis. If the flow rate of the coolant is not known, you can use the Reynolds number to define the flow rate.

This topic provides you with some issues to consider when deciding on a coolant control method.

Coolant control issues

The mold designer should know what the available coolant flow rate is before designing the mold. The mold design can be modified in the early stages to take account of the availability of coolant at the plant.

In the cooling circuits, the cooling medium must absorb the heat by convection. This means that the flow must be turbulent. The power required to pump coolant around the system is proportional to the cube of the flow rate. This means that doubling the water flow requires eight times the pumping power. The use of excessively high flow rates is a costly waste of energy.

Coolant control options

The following options are available:

Specified pressure. A total pressure for the cooling circuit. The Cool analysis determines the corresponding flow rate using the physical properties of the coolant and the coolant temperature.

Specified flow rate. The flow rate of coolant that can be set on the molding machine.

Specified Reynolds number. When you do not know the flow rate or the total pressure through the circuit, and want the Cool analysis to determine a suitable flow rate, based on a minimum Reynolds number to be achieved in the cooling circuit. The recommended setting is 10,000.

> **NOTE:** When using parallel cooling circuits, you may not be able to achieve the required Reynolds number in all branches.

Total Flow-rate (All Circuits). When the individual circuit pressure or flow rates are not known, and you want to specify the total flow rate through the cooling system. The total flow rate is divided among the circuits according to their pressure drops. Circuits with a low pressure drop receive more of the flow rate, and circuits with a higher pressure drop receive less.

> **NOTE:** If you choose this option, ensure you specify the same total flow rate value for all coolant inlets in the model.

Reynolds number

Once turbulent flow has been achieved, an increase in flow rate makes little difference to the rate of heat extraction. Therefore, set the flow rate only to achieve the ideal Reynolds Number with minimum variations. The Reynolds Number is a ratio that defines the rate of fluid flow and is assigned to cooling circuits when the flow rate is not known.

When running an analysis, we recommend using a Reynolds number of 10,000, however, to represent turbulent flow, then check this result to ensure minimum variation. Don't aim for a Reynolds number greater than 10,000.

If you have parallel cooling channel circuits, it may be difficult to achieve minimum variation of the Reynolds number throughout all the branches.

If this is the case, consider changing the circuit layout. If there is a large variation in cooling channel diameter, there may be excessive variations in the Reynolds number. If this occurs, either adjust the cooling channel diameter, or reduce the minimum Reynolds number (ensure the Reynolds number is always greater than 4000, which is considered fully turbulent).

Effect of parallel cooling circuits on flow rate

The following problems can arise when using parallel cooling channels:

- The flow rate in each branch is reduced as extra branches are added. This reduces cooling efficiency, unless the total flow rate is increased.
- The flow rate can vary from one branch to another, causing non-uniform cooling. To minimize this, you can adjust the bore of the varying branches to balance the coolant flow.
- If one branch gets partially blocked by debris, the flow rate in that branch can be dramatically reduced while increasing slightly in the other branches. This also causes non-uniform cooling.

Cooling circuit parameters

To control the coolant in the cooling circuits (channels), you can either specify a coolant pressure, coolant flow rate, or a Reynolds number.

Editing cooling circuit parameters

To control the coolant in the cooling circuits (channels), you can either specify a coolant pressure, coolant flow rate, or a Reynolds number.

If the flow rate of the coolant is not known, you can use the Reynolds number to define the flow rate.

NOTE: This topic describes setting coolant control for an existing coolant inlet. You can also set coolant control when setting a new coolant inlet.

---

1. Right-click the arrow on the inlet of the cooling circuit you want to adjust and select **Properties** from the menu that appears.
   The **Coolant inlet** dialog appears.
2. Select the parameters of the coolant.

   NOTE: The **Coolant control** drop-down list allows you to specify the coolant properties based on coolant pressure, flow rate, or Reynolds number.

   ---

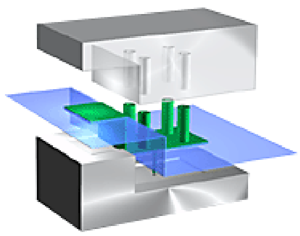
3. Click **OK**

Cooling planes

A cooling plane needs to be created before you can model any part of a cooling circuit.

Cooling planes are always created parallel to the X-Y plane.

Since a mold must contain both a cavity and core, there may be a small gap between them. While this gap is too small for material to flow into, it still acts as a partial barrier to heat transfer. To simulate this effect, you need to model the surface between the cavity and core as a parting plane.



A parting plane represents the resistance of heat transfer across a surface inside a mold, usually where the two halves of the mold meet. By default, the solver assumes perfect conductivity between mating surfaces. Thermal interface conductance is the only attribute that can be specified as a parting plane property. If a parting plane has low contact pressure or a small gap, and it is located between the part and the cooling channels, it should be modeled.

TIP: Click **Bottom View** to display the model in the best orientation for working with cooling planes.

Cooling planes

Cooling planes are used to help model the cooling circuit.

Modeling cooling planes

For an accurate analysis, a cooling plane must be modeled correctly.

1 Create regions which represents the plane(s) where the mold cavity and core meet, using the **Geometry tab > Create panel > Regions** tools. The region(s) should completely surround the part and have a hole where the part is.

2 Select all the regions you just created.

3 Click **Geometry tab > Properties panel > Edit**.

4 Click **New > Parting surface**.

The **Parting Surface** dialog appears.

5 Specify the interface conductance across the parting plane and click **OK**.

The interface conductance depends on the mold material, the contact pressure, whether there is air or grease between the mold cavity and core, and the thickness of the gap (if any).

6 Mesh the new regions by clicking **Mesh tab > Mesh panel > Generate Mesh**.

7 If you have not done so already, model the external mold surface.

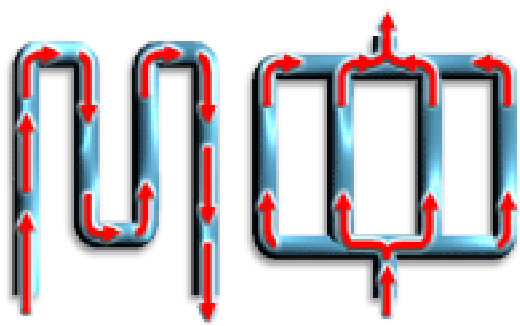
Leave a small gap between the edge of the parting plane and the mold surface to avoid element intersection problems.

TIP: A parting plane can be placed at any location within the mold to view the thermal distribution across that plane.

Parallel and series cooling circuits

Cooling circuits are generally classified as series or parallel circuits. In both types of circuits, the final temperature rise of the coolant is determined entirely by the energy input from the plastic and the volumetric flow rate of the coolant.

The most important factors in maintaining effective heat transfer are, therefore, flow rate and circuit design. The following diagram shows a series circuit on the left and a parallel circuit on the right.



It is easier to control coolant velocity in a series circuit because the flow rate in each section is the same. It is therefore easier to maintain flow rate conditions that provide effective heat transfer.

Parallel circuits

Parallel cooling channels are drilled straight through from a supply manifold to a collection manifold. Due to the flow characteristics of the parallel design, the flow rate along each of the cooling channels may be different, depending on the flow resistance of each individual cooling channel. These varying flow rates in turn result in different heat transfer efficiency for each cooling channel. As a result, cooling may not be uniform with a parallel circuit.

Typically, the cavity and core sides of the mold each have their own parallel circuit. The number of cooling channels per circuit varies with the size and complexity of the mold.

Only use a parallel circuit if your model has one or more of the following circumstances:

- The pressure drop over a series circuit is too high to be realistic.
- An area of the mold cannot be effectively cooled with a series circuit.
- You are simulating the manifold carrying coolant to the mold.

When a parallel circuit is used, each branch must be capable of extracting the heat load from the surrounding area. Coolant flow must be regulated by specifying the diameter and length of each branch within the circuit.

Each branch should have turbulent flow to give an effective heat transfer coefficient. The surface area of the branch is determined by balancing its length and diameter against the localized heat load.

A balanced parallel circuit provides uniform heat extraction; however, parallel circuits also have the following disadvantages:

- The flow rate in each branch is reduced when extra branches are incorporated. This reduces cooling efficiency unless the total flow rate is increased accordingly.
- Each cooling channel may have a different flow rate, causing non-uniform cooling. This disadvantage can be minimized by adjusting the diameter of the branches to balance the coolant flow.
- If one branch is partially blocked by debris, the flow rate in that branch may be dramatically reduced while the flow rate may slightly increase in other branches. This will cause non-uniform cooling.

Series circuits

Cooling channels connected in a single loop from the coolant inlet to the outlet are called serial circuits. This is the most common type of cooling channel. If the cooling channels are uniform in size, the coolant can maintain its turbulent flow rate through the entire circuit.

The coolant will continue to collect heat along the cooling circuit so you should ensure that the temperature rise of the coolant from inlet to outletis minimized. The temperature difference of the coolant at the inlet and the outlet should be within 5°C for general purpose molds, and within 3°C for precision molds. More than one serial circuit may be required for large molds to ensure uniform coolant temperature and cooling.

Due to the problems experienced with parallel circuits, series circuits are generally preferred, but these cannot always be used. Series circuits should not be used in the following situations:

- The length of the series circuit results in a pressure drop that is too high for the available pump capacity.
- Physical constraints in the mold design mean that the mold cannot be effectively cooled with a series circuit.

Cooling circuit design references

The following references provide additional information to help you design efficient cooling circuits.

- Gastrow, Hans *Injection Molds: 108 Proven Designs* Edited by K. Stoeckhert. Hanser Publishers, Munich Vienna New York 1993.
- Menges, G. and Mohren, P. *How To Make Injection Molds, 2nd Edition* Hanser Publishers, Munich Vienna New York 1992.
- Douglas M. Bryce. *Plastic Injection Molding: Mold Design and Construction Fundamentals* Society of Mechanical Engineers 1998.

Cooling components

5

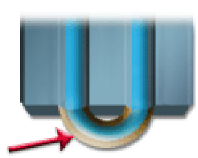
In addition to the cooling channels, there are several other components that make up a cooling circuit, such as hoses, coolant inlets bubblers and baffles. Some are required while others are optional, depending upon the complexity of the model.

Hoses

Hoses connect the cooling channels and enable the flow of coolant through the cooling circuit.

Hoses contribute only to the pressure drop of the coolant flow, and not the heat transfer calculation.

Two cooling channels of different diameters can be connected by hoses. The following diagram shows two cooling channels connected by a hose.



Hoses are modeled as beam elements and are defined by two end points and a diameter.

Hoses

This dialog is used to edit the properties of the hoses.

To access this dialog, select the hose, right-click and select **Properties**

Hose dialog

This dialog is used to edit the properties of **Hoses**.

To access this dialog to edit the properties of existing model entities, select at least one **Hose**, then either select **Geometry tab > Properties panel > Edit**, or press **Alt-Enter**, or right-click and select **Properties**.

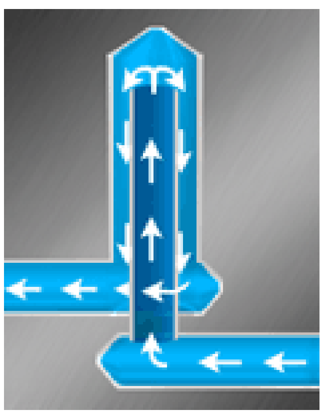
The set of property values defined by the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

NOTE: Always use the Hose property rather than the Connector property when modeling cooling channels.

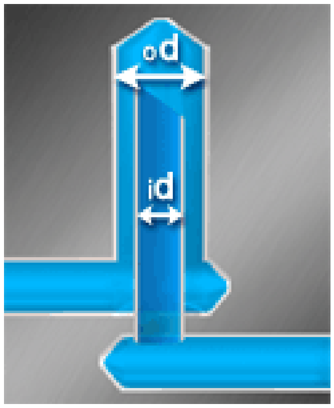
Bubblers

Areas of the mold which cannot be cooled effectively by normal cooling channels may require the use of bubblers. Bubblers divert the coolant flow into areas that would normally lack cooling.

Bubblers are created by fitting a tube in the center of a drilled hole, forming an annular channel. The coolant flows into the bottom of the tube and bubbles out of the top, like a fountain. The coolant continues down around the outside of the tube and continues through the cooling channels. The following diagram illustrates the coolant flow through a bubbler.



Bubblers provide the most effective cooling of parts with slender cores. The diameter of both sections of the bubbler must ensure that the flow resistance in both sections is equal. The condition for this is—inner diameter / outer diameter = 0.707



Bubblers are commercially available and are usually screwed into the core. The tubing should be beveled at the end to enlarge the cross section of the outlet for diameters up to 4mm. Bubblers can be used for core cooling and for cooling flat mold sections where drilled or milled channels can not be used.

NOTE: Because bubblers have narrowed flow areas, the flow resistance increases. Therefore, care should be taken in designing the size of these devices. The flow and heat transfer behavior can be readily modeled and analyzed by Autodesk Moldflow.

Bubblers

Placement of bubblers in a mold assist in the management of localized hot spots in the mold.

Modeling bubblers

Bubblers are created within a mold by drilling a blind hole and then fitting a smaller diameter concentric tube up the hole almost to its top, thus creating a central circular channel and an outer annular channel. Coolant is introduced into the inner channel and flows along the channel to the top of the bubbler hole, returning along the outer channel to the outlet point. By this means cooling can be introduced to otherwise inaccessible areas of the mold.

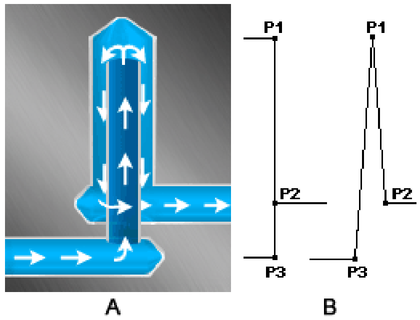


Figure 3: A indicates the bubbler cross-section and B indicates the mold model representation. The beam element between P3 and P1 is circular in cross-section with Heat Transfer Effectiveness (HTE)=0. The beam element between P1 and P2 is annular in cross section with HTE=1. Orange is the default color assigned to a bubbler.

1. Click **Geometry tab > Create panel > Curves > Create Line** and create a line from the center line of the inlet (P3), to the top of the tube (P1).
2. Create a line from the top of the tube (P1), to the center line of the outlet hole (P2), including a small distance along the axis of the bubbler hole to separate the two new curves.
3. Select the inner channel (line P1-P3), right-click and select **Properties**. Click **Yes** to the **Assign new property?** prompt.
4. Click **New** and select **Channel**.
5. Enter **0** for the **Channel heat transfer effectiveness** (HTE).
6. In the **Name** box, enter an appropriate name for the channel property, then click **OK** twice.
7. Select the outer channel (P1-P2), right-click and select **Properties**. Click **Yes** to the **Assign new property?** prompt.
8. Click **New** and select **Bubbler**.
9. Enter **1** for the **Channel heat transfer effectiveness** (HTE).
10. Specify the required dimensions, mold material, name the bubbler appropriately, then click **OK** twice.

    The inner diameter of the bubbler should be equal to or larger than that of the channel you created in **Step 4**.
11. Right-click on the **Mesh Type Selection** task in the Study Tasks pane and select **Define Mesh Density**. Set the mesh density to 2.5 times the diameter of your bubbler. Click **Apply** and **OK**.
12. Click **Mesh tab > Mesh panel > Generate Mesh**, deselect the check boxes, then click **Mesh Now**.

NOTE: You should leave a gap of at least half of the diameter of the bubbler between the top of the bubbler and your part. This is to allow clearance for the dome at the top of the bubbler.

TIP: You may overlay the two curves if you prefer. If so, it is recommended that during **Step 2**, you create the second curve (P1-P2) away from the part, assign its properties, then move it so that it is superimposed on the curve (P3-P1).

Correcting bubblers imported from C-MOLD 2000

In Autodesk Moldflow Insight, bubblers must be modeled as two beam elements, one to describe the flow of coolant into the bubbler, and the other to describe the flow out of the bubbler.

Bubblers imported from C-Mold 2000 consist of only one beam element and will not be treated correctly by the cooling solver in Autodesk Moldflow Insight. To correct this problem, please follow the steps below.

NOTE: You may also delete the bubbler imported from C-Mold and build a new bubbler according to the instructions provided in the online help for modeling a bubbler.

In Autodesk Moldflow Insight, a bubbler is modeled using an inner channel element with HTE=0, and an outer bubbler element with HTE=1. Orange is the default color assigned to a bubbler.

1. In the **Layers** pane, turn off all layers other than the one to which the bubbler was assigned when it was imported.
2. Click on the bubbler to select it and then click **Geometry tab > Properties panel > Assign** to display the **Assign Property** dialog.
3. Click **New** and select **Channel** from the drop-down list.
4. In the **Channel** dialog, specify the inner diameter of the bubbler, assign a heat transfer effectiveness value of **0**, and then click **OK** twice to apply the new properties.
5. Click **Geometry tab > Utilities panel > Move > Translate**.
6. In the model pane, click on the element representing the inner channel of the bubbler.
7. Select **Copy** and specify a vector that will move the new element away from the part. Click **Apply**.

   This will create a copy of the element, which you can easily select and change its properties.
8. Click on the new element to select it and then click **Geometry tab > Properties panel > Assign**.
9. Click **New** and select **Bubbler** from the drop-down list.
10. In the **Bubbler** dialog, specify the inner and outer diameters of the bubbler, assign a heat transfer effectiveness value of **1**, and then click **OK** twice to apply the new properties.

The inner diameter of the bubbler must be equal to or greater than that of the inner channel.

11 Click **Geometry tab > Utilities panel > Move > Translate**.

12 In the graphics pane, click on the new bubbler you have made.

13 In the **Move/Copy—Translate** dialog, reverse the sign of each of the vector components you specified earlier and click **Apply**.

This will move the element so that the bubbler is superimposed on the channel element.

14 Click **Mesh tab > Mesh panel > Density**. Set the mesh density to 2.5 times the diameter of your bubbler. Click **Apply** and **OK**.

15 Click **Mesh tab > Mesh panel > Generate Mesh**, deselect the check boxes, and click **Mesh Now**.

NOTE: You should leave a gap of at least half the diameter of the bubbler between the top of the bubbler and your part. This is to allow clearance for the dome, which forms the top of the bubbler.

Bubblers

These dialogs are used to edit the properties of the bubblers.

To access these dialogs, select the bubbler, right-click and select **Properties** from the drop-down menu.

Bubbler dialog—Bubbler Surface Properties tab

The **Bubbler Surface Properties** tab of the **Bubbler** dialog is used to specify the geometrical and thermal properties of the bubbler.

Bubbler dialog—Mold Properties tab

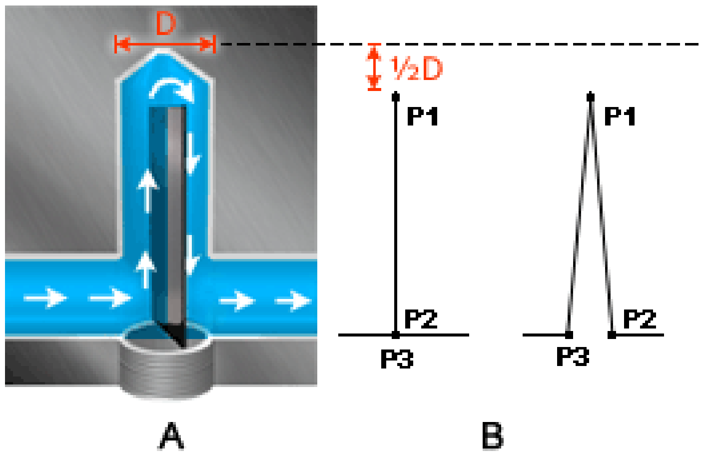
The **Mold Properties** tab of the **Bubbler** dialog is used to specify the properties of the mold block containing the bubbler.

Baffles

Areas of the mold which cannot be cooled effectively by normal cooling channels may require the use of baffles. Baffles divert the coolant flow into areas that would normally lack cooling.

A baffle is a cooling system component which is constructed by inserting a metal plate in the cooling channels. The plate forces the coolant to flow up one side of the baffle and down the other. The baffle interrupts the flow in the cooling channels creating turbulence around bends which improves the heat transfer capability of the coolant.

Baffles are modeled by creating two circular beam elements with a gap at the top for the coolant to flow around. The first circular element represents the flow up one side of the baffle and the second element is for the flow down the other side. Yellow is the default color assigned to a baffle. In the following diagram, **A** indicates the baffle cross-section and **B** indicates the mold model representation. The baffle is made up of two circular beam elements with HTE (Heat Transfer Effectiveness)=0.5.



NOTE: Because baffles have narrowed flow areas, the flow resistance increases. Therefore, care should be taken in designing the size of these devices. The flow and heat transfer behavior can be readily modeled and analyzed by Autodesk Moldflow.

Baffles

Placement of baffles in a mold assist in the management of localized hot spots in the mold.

Modeling baffles

A baffle is a cooling system component which is constructed by inserting a metal plate in the cooling channels. The plate forces the coolant to flow up one side of the baffle and down the other.

1. Click **Geometry tab > Create panel > Curves > Create Line** and create a line from the center line of the inlet (P3), to the top of the tube (P1).

   NOTE: The position of P1 must be at least one half of the diameter of the baffle shorter than the total height of the baffle. This allows space for the flow of cooling fluid over the baffle.

   ---

2. Create a line from the top of the tube (P1), to the center line of the outlet hole (P2), including a small distance along the cooling channel to separate the inner and outer channels.

Option: You may overlay the two curves if you prefer.

3 Select the inner channel (line P1-P3), right-click and select **Properties**. Click **Yes** to the **Assign new property?** prompt.

4 Click **New** and select **Baffle**.

5 Enter **0.5** for the **Channel heat transfer effectiveness** (HTE).

6 Specify the required dimensions and mold material. In the **Name** box, enter an appropriate name for the channel property, and click **OK** twice.

7 Select the outer channel (P1-P2), right-click and select **Properties**. Click **Yes** to the **Assign new property?** prompt.

8 Select the baffle you named in **Step 6** and click **OK**.

9 Right-click on the **Mesh Type Selection** task in the Study Tasks pane, and select **Define Mesh Density**. Set the mesh density to 2.5 times the diameter of your baffle. Click **Apply** and **OK**.

10 Click **Mesh tab > Mesh panel > Generate Mesh**, deselect the check boxes and click **Mesh Now** .

NOTE: If the top of the baffle will be close to your part, you should leave a gap of at least one diameter of the baffle between the top of the baffle and your part. This is to allow clearance for the dome which forms the top of the baffle.

Correcting baffles imported from C-MOLD 2000

In Autodesk Moldflow Insight, baffles must be modeled as two beam elements, one to describe the flow of coolant up the baffle, and the other to describe the flow down the baffle.

Baffles imported from C-Mold 2000 consist of only one beam element and will not be treated correctly by the Cool solver in Autodesk Moldflow Insight. To correct this problem, please follow the steps below.

NOTE: You may also delete the baffle imported from C-Mold and build a new baffle according to the instructions provided in the online help for modeling a baffle.

In Autodesk Moldflow Insight, a baffle is modeled using two semi-circular elements with Heat Transfer Effectiveness=0.5. Yellow is the default color assigned to a baffle.

1 In the **Layers** pane, turn off all layers other than the one to which the baffle was assigned when it was imported.

2 Select the baffle and then click **Geometry tab > Properties panel > Edit**.

Ensure that the baffle has been assigned the property **Baffle** and that it has a Heat Transfer Effectiveness=**0.5**.

3 Click **Geometry tab > Utilities panel > Move > Translate**.

4 In the model pane, click on the element representing the baffle.

5 Select **Copy**, specify the vector **0 , 0 , 0** and click **Apply**.

This will create a copy of the element, superimposed on top of the existing one.

6 Click **Mesh tab > Mesh panel > Density**. Set the mesh density to 2.5 times the diameter of your baffle. Click **Apply** and **OK**.

7 Click **Mesh tab > Mesh panel > Generate Mesh**, deselect the check boxes and then click **Mesh Now**.

NOTE: You should leave a gap of at least half the diameter of the baffle between the top of the baffle and your part. This is to allow clearance for the dome, which forms the top of the baffle.

Baffles

These dialogs are used to edit the properties of your baffles.

To access these dialogs, select the baffle, right-click and select **Properties** from the drop-down menu.

Baffle dialog—Baffle Surface Properties tab

The **Baffle Surface Properties** tab of the **Baffle** dialog is used to specify the geometrical and thermal properties of the baffle.

Baffle dialog—Mold Properties tab

The **Mold Properties** tab of the **Baffle** dialog is used to specify the properties of the mold block containing the baffle.

Flow control valves

Molds often have many individual cooling circuits which could require individual controllers to obtain optimum heat transfer.

Since one mold temperature control unit per mold is common practice, it is not possible to control the individual circuits by using variable inlet temperatures. With flow control valves, each cooling circuit can have a different flow rate. This enables you to optimize the heat extracted from different parts of the mold.

Coolant

6

A coolant is a fluid which flows through the cooling channels to regulate the temperature of the molten plastic in the mold. An ideal coolant has high thermal capacity, low viscosity, is low-cost, and is chemically inert, neither causing nor promoting corrosion of the cooling system.

Common coolants

A cooling system uses water or another cooling medium to flow through the cooling circuits of the mold to control the temperature of the metal surfaces that come in contact with the plastic being molded.

Commonly used coolants

The following table lists the most commonly used coolants:

Coolant	Temperature
Ordinary water from mains or a cooling tower	20°C–25°C
Ordinary water from a heater/circulator unit	> 30°C
Cold water from a chiller	> 10°C
Water mixed with antifreeze from a chiller	> 5°C
Oil—from a heater/circulator unit	> 80°C

An important parameter that the mold designer must know is the pressure of coolant available at the plant. If the available pressure is less than the pressure that the cooling channels require, the cooling of the part could be ineffective due to non-turbulent flow of coolant.

Choosing a coolant

Adding glycol (antifreeze) to the coolant will raise its viscosity, resulting in higher pumping pressures being required, or a lower flow rate. Using a chilled water and glycol mix may be less effective cooling than a well-designed circuit with plain tower water circulating at the optimum flow rate. Oil is generally used only when very high mold temperatures are required. The available pump capacity must be greater than that required by the cooling circuits in the mold.

Common coolants

Use these dialog to specify properties for coolants.

Coolants dialog

This dialog is used to view/edit the properties of the selected coolant.

The collection of property values defined on the dialog are saved to a property set with the description shown in the **Name** box.

Coolant flow

Coolant flow behaviour affects heat transfer between the mold and coolant. Heat transfer effectiveness is increased when the coolant flow is turbulent and not laminar. Turbulent coolant flow has a better temperature gradient than laminar flow.

The temperature gradient from the cavity to the cooling channel has two components:

- The temperature gradient across the metal, which depends on the conductivity of the metal
- The temperature gradient across the coolant and metal interface, which depends on coolant flow

The heat flow path from plastic to cooling channel

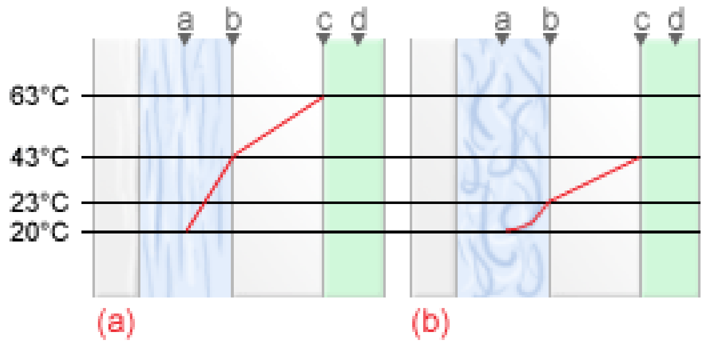
The speed of coolant flow can affect the heat transfer.

At very low speeds the flow is laminar; the heat has to be conducted through various layers of coolant to the center of the channel. Since coolant is a poor conductor of heat, heat transfer is very inefficient. This is shown in the following graph **a** below where there is a relatively large temperature difference between the coolant metal interface and the center of the channel.

As coolant flow increases, heat transfer increases at a marginal rate until the coolant flow becomes turbulent. There is now a component of coolant velocity perpendicular to the channel which causes a dramatic improvement in heat transfer. The greater heat transfer shown in following graph **b** results in a lower temperature at the cavity wall for coolants with a turbulent flow.

In these graphs, the mold elements are represented by the following:

a - Coolant
b - Water/metal interface
c - Cavity wall
d - Plastic part



Laminar and turbulent flow

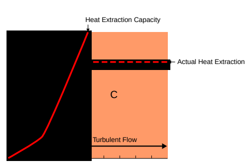
The relationship between heat transfer and coolant flow can be expressed as a power factor. Coolant flow is either laminar or turbulent, or in transition between laminar and turbulent. For laminar flow, heat transfer increases proportional to the cube root of the flow rate.

This means that doubling the coolant flow increases the heat transfer by about 24 percent. For fully turbulent flow, the heat transfer is proportional to the square of the cube root of the flow rate. Therefore, in the turbulent zone, doubling the coolant flow increases heat transfer by 59 percent.

The power required to pump coolant around the system is proportional to the flow rate cubed. This means that doubling the coolant flow will require eight times the pumping power.

When turbulent flow has been fully developed, greater increases in flow rate will increase the molds capacity to extract heat. However, the amount of heat that can be extracted is limited by the amount of heat passing into the mold, and heat extraction may not be improved beyond this limitation.

The most effective condition for heat transfer is to ensure that coolant flow is turbulent, and the capacity to extract heat does not greatly exceed the amount of heat available for extraction. The following graph shows heat extraction from the mold as a function of the coolant flow rate. In region **A** heat transfer is by conduction, in region **C** heat transfer is by convection, while **B** is a region transition.



Reynolds number

The Reynolds number is a ratio that defines the rate of fluid flow in pipes. The onset of turbulence in water is between 2300-4000. The Reynolds number is assigned to cooling circuits when the flow rate is not known. A Reynolds number of 4000 is considered fully turbulent, but it is good practice to use a Reynolds number of 10,000 to represent turbulent flow when running an analysis.

Cooling system equations have more information about the Reynolds Number.

Typical coolant temperatures

When the mold temperature for a part has been established, an appropriate coolant can be selected. Coolant temperatures are typically 10ºC cooler than the mold surface.

The temperature range for commonly used coolants is listed in the following table.

Coolant	Temperature
Ordinary water from mains or a cooling tower	20 to 25°C
Cold water from a chiller	above 10°C
Water mixed with antifreeze (usually glycol) from a chiller	-5°C
Oil—normally from a heater/circulator unit	80°C or above

Adding glycol to the coolant will raise its viscosity, resulting in a higher pumping pressure being required, or a lower flow rate. Using a chilled water and glycol mix may be less effective than a well designed cooling circuit with plain tower water circulating at the optimum flow rate. Oil is used only when very high mold temperatures are required.

Determining the coolant temperature

The coolant temperature is determined by considering the recommended mold surface temperature for the plastic being molded and the feasibility of achieving this temperature.

Some thermoplastics need a higher surface temperature for better part finish. In such cases, heated water or oil may be used for cooling the mold. Generally the temperature of the coolant is lowered to achieve better heat transfer and reduced cycle time. However, try to keep the mold surface temperature close to the recommended value.

An inlet temperature 10 to 20°C lower than the desired mold temperature is suggested as a design guide. This will also be influenced by the coolant circulating plant available. There is no benefit specifying a coolant inlet temperature of 5°C if no chiller is available to achieve this value. The tower water temperature will vary according to its geographical location and ambient conditions.

Coolant inlets

The inlet is the location on the mold where the coolant enters the cooling circuit.

You must set the inlet location before running a Cool analysis. The coolant type and the coolant inlet temperature can be specified when you set the inlet location.

Coolant inlets

Coolant and coolant flow parameters can be defined for an analysis.

Setting coolant inlets

To run a Cool analysis, you must specify the location that coolant enters the cooling circuit (channel) in your mold design.

The coolant type and coolant inlet temperature can also be specified while selecting the coolant inlet location, so that the analysis can calculate the heat extraction efficiency of your mold design.

1. Ensure that you have a Cool analysis specified, and the cooling circuit (channel) displayed on screen.
2. Click **Boundary Conditions tab > Cooling panel > Cooling Inlets**.
3. Click **New** in the **Set Coolant Inlet** dialog.

   The **Coolant Inlet** dialog appears, allowing you to select a coolant and coolant control option.
4. Click **Select** next to the **Coolant** drop-down list, and choose the required mold coolant.
5. From the **Coolant Control** drop-down list, select a coolant control option. Specify the coolant control by either pressure (default), flow rate, or Reynolds number, and edit the default value, if necessary.
6. In the **Coolant Inlet temperature** box, enter the required coolant inlet temperature.
   If required, enter a new **Name** in the name box.
7. Click **OK**.
8. Now, click the cross-hairs on the cooling channel where coolant will enter.

   The model will automatically update with a blue arrow and C, indicating that the coolant inlet is successfully set.
9. Right-click the mouse and select **Finish Coolant Inlets**.

Coolant inlets

This dialog is used to set the properties of cooling inlets.

To access this dialog, select the cooling inlet, right-click and select **Properties** from the drop down menu.

Coolant Inlet dialog

This dialog is used to set the properties associated with a coolant inlet.

To access this dialog, select a coolant inlet symbol in the model pane, then either select **Geometry tab > Properties panel > Edit**, or press **Alt-Enter**, or right-click and select **Properties**.

Coolant inlet temperature

The temperature of the coolant as it enters the cooling system is an important aspect to consider for a Cool analysis.

To achieve effective heat extraction from the part, the coolant temperature should not increase by more than 2–3°C from inlet to outlet. A greater increase may indicate a problem with the cooling system design.

The Cool analysis log records the change in coolant temperature from inlet to outlet. If the increase is unacceptable, that is greater than 2–3°C, you can use the coolant temperature result to determine where the greatest increase in temperature occurs. If necessary, the inlet temperature and the degree of turbulence in the cooling channels can be altered to increase heat extraction in areas that are difficult to cool.

Ideal inlet temperature

The temperature of the coolant should be as low as possible, in order to achieve the best heat transfer and the lowest cycle time. However, some thermoplastics need a higher surface temperature for better part finish. Heated water or oil may be used for cooling the mold in these cases. Try to keep the mold surface temperature close to the recommended value.

Available inlet temperature

In general, use an inlet temperature of 10–20°C lower than the desired mold temperature. However, you should know what temperature coolant the coolant circulating plant is able to provide. There is no benefit specifying a coolant inlet temperature of 5°C, if no chiller is available to achieve this value. Also, the tower water temperature will vary depending on geographical location and ambient conditions. The inlets and outlets should ideally always be positioned on the bottom of the mold, to eliminate the risk of coolant dripping onto the mold faces.

Coolant inlet temperatures

Once you have set the coolant inlets, you can specify and change the coolant properties at any time.

Changing the temperature of coolant inlets

The procedure below can be used to set or change the temperature of single or multiple coolant inlets.

1. Select the coolant inlet.

   To select multiple inlets, hold down the **Ctrl** key and click on the required inlets.
2. Right-click on the coolant inlet and select **Properties** from the context-menu.

   The **Coolant inlet properties** pane appears.
3. Change the properties of the coolant inlet as needed.
4. Click **OK**.

Pump capacity

The pump capacity available at the plant must be greater than that required by the cooling circuits in the mold.

The amount of heat which has to be removed can be calculated from the weight of the part and its enthalpy at molding temperature, allowing for the fact that the temperature of the part when ejected will be generally well above ambient.

Enthalpy can be defined by the formula:

$$\mathbf{H = U + pV}$$

where:

H is the enthalpy
U is the internal energy of the system
p is pressure
V is volume

Use of tower water

Individual water temperature controllers are not available in some manufacturing plants, so the coolant supply comes from another source, which is usually an external re-circulating water cooling tower.

Coolant derived from such a source is typically called tower water.

In these cases, the inlet temperature is fixed so this fixed variable has to be taken into account when designing the optimum cooling circuits and cooling conditions.

Fouling of water lines

When the mold is in production, deposits are likely to form on the cooling channel surfaces, especially in hard water areas. This is called fouling and it greatly reduces the effectiveness of heat transfer from the metal to the coolant.

The thermal conductivity of mineral can be 98 percent less than tool steel so 1 mm of fouling deposit gives the equivalent thermal resistance to 50 mm of tool steel.

Fouling is a production maintenance problem, and the effect of fouling should not be considered at the design stage.