Autodesk® Moldflow Insight 2019

Coolant Flow Solver

Executive summary

Existing Autodesk Moldflow customers have requested a standalone channel flow module based on the Autodesk Moldflow one dimensional Cooling channel solver that can be used to simulate the cooling channels in the mold as well as the surrounding hoses of the mold.

The aim of the Coolant flow solver is to allow users to optimize the heat exchange in the mold through careful design of the cooling channels without the requirement of having to model the part together with the mold. The users want to be able to focus on the cooling system design without the overhead of the part and mold models in the analysis that take a long time to model and solve.

The feature allows the user to focus solely on the cooling system of the mold without the part and the mold present. It gives the users more control over the selection of the minor loss factors and pipe friction loss formulations that can be used in an analysis. Users also have the option of taking gravity into account and the option of using a specific pump curve to model the pump in the analysis is supported. Specific thermal boundary conditions can be set on specific elements to evaluate the heat removing capability of the mold circuit design.

This module has introduced new processes, elements and process condition options. All these options and results have been carried through to the existing Cool and Cool (FEM) analyses, as well as the midplane, dual domain and 3D mesh options.



Contents

Introduction **3**
Channel and Pipe flow theory **5**
Computational Domain 7
Discretization 8
Head Loss formulations 9
Darcy-Weisbach formulation 10
Minor Losses 14
General pressure losses 15
Elevation 17
Pumps 18
Simulation implementation **23**
Boundary conditions 25
Flow boundary conditions 25
Thermal boundary conditions 26
Elevation 27
Elemental Properties **27**
Pipe or channel elements 27
Pump elements 28
Coolant flow modelling in Moldflow. **29**
Process Settings 30
Element options 33
Pump elements 35
Coolant Flow result type **38**
Results **39**
Conclusions **45**
References **46**



Introduction

Cooling channels can make up complex flow systems that could involve numerous parallel and series connections with baffles and bubblers placed at varying heights in an injection molding shop. The engineering objective in such complex cooling systems is to ensure that fluid can be delivered to a point at the injection molding machine at a specified flow rate and pressure at minimal cost. In complex cooling systems cost considerations include the initial capital outlay as well the operational and maintenance costs. To contain costs in the coolant flow system, prudent selection of pipe diameters need to be made in conjunction with the pumps chosen to drive the fluid through the cooling system.

These cooling systems typically involve several pipes connected to each other in series or in parallel. With series connection the mass flow rate through the entire system remains constant regardless of the individual diameters or cross sectional shapes of the system. This is a natural consequence of the conservation of mass principle for steady flow. The total pressure loss is equal to the sum of all the pressure losses in the individual components in the series system. In large systems the main contributor to pressure loss are frictional losses in the system. In addition to frictional losses are minor losses caused by the fluid flowing through connectors, valves, bends, elbows, tees, inlets, exits, enlargements, and contractions etc.

For a cooling system, figure 1, that has channels branching out from a common junction, junction A, into two or more parallel sections, branches 1,2 and 3, that rejoin at a common junction further downstream, junction B, the total flow rate is the sum of the flow rates in the individual sections 1, 2 and 3. The pressure losses in each individual branch connected by the same junctions, A and B, in parallel must be same. Therefore, the relative flow rates in parallel branches are established from the requirement that the pressure loss in each branch be the same.

From figure 1 the pressure loss through branches 1 and 3 must be the same as the pressure loss through branch 2. However, the distance the fluid must travel to flow through branches 1 and 3 is much greater than the flow distance through branch 2. The flow will always follow the path of least resistance; hence it is obvious that for the same pressure loss more flow will flow through branch 2 than will flow through branches 1 and 3. The total flow through the system will be the sum of the flow through branches 1, 2 and 3.

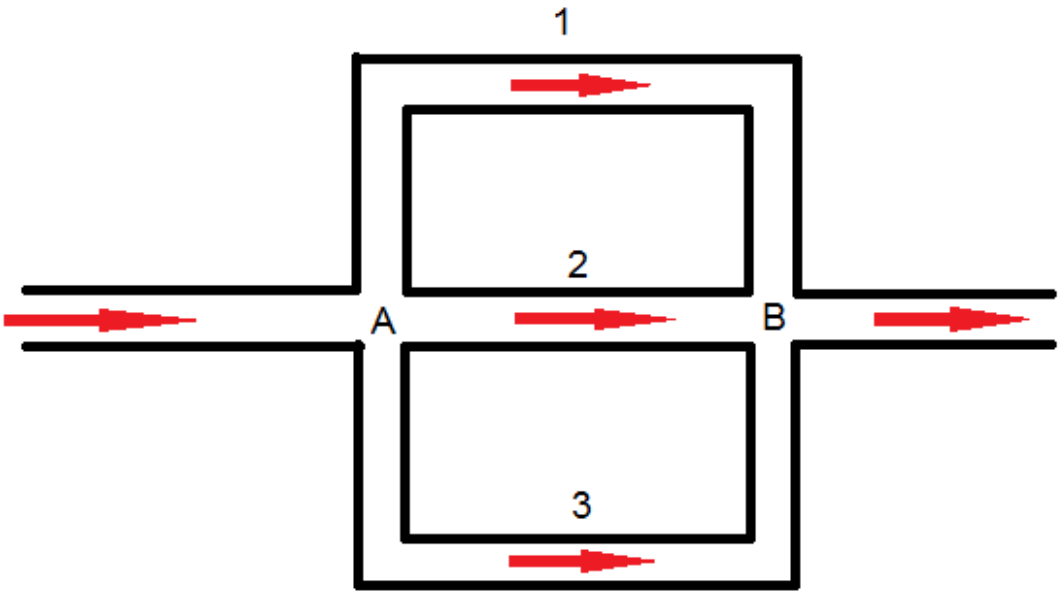


Figure 1 Parallel flow through a system.

From figure 1 it can be concluded that the head loss is the same in each branch and that the total flow rate is the sum of the flow rates in all the individual pipes or channels.

Therefore, the analysis of systems, no matter how complex they are, is based on two simple principles.

1. *Conservation of mass throughout the system must be satisfied.* This is commonly known as the continuity principle. This is satisfied by ensuring the total flow into a junction is equal to the total flow out of the junction for all the junctions in the system. In addition to this, the flow rate must remain constant in the channels connected in series, regardless of the changes in diameters.
2. *Pressure drop between two junctions must be the same for all paths between the two junctions.* This known as the work energy principle.

Hence the analysis of flow in cooling systems is very similar to the analysis of electric circuits with the flow rate corresponding to electric current and pressure to electric potential. However, the flow circuit is more complex than the electric circuit in that the “flow

resistance" is a highly nonlinear function. Therefore, the flow circuit requires the solution of a system of nonlinear equations to be solved simultaneously. The solutions of these equations are very complex and numerous algorithms have been proposed for the solution of these equations.

Popular solving methods fall into node, loop or element equations that are being solved. In the past loop solving methods have been popular due to them requiring less computer storage than node or element methods. However, loop methods require an initial distribution that satisfies continuity which is difficult to ascertain. Element based methods do not require initial condition flows that satisfy continuity and converge very fast, however they require more computer storage than node and loop methods. They also require the loops to be specified. Methods based on node equations require less computer storage than element methods and do not require the loops to be specified nor do they require accurate initial conditions. They can also deal with mixed boundary conditions. That is flow and pressure boundary conditions can be specified separately in the system. Node based methods are used in Computational Fluid Dynamics (CFD) codes. An example of this is the SIMPLE method proposed by Patankar[1]. Greyvenstein and Laurie showed that node based methods using the SIMPLE method work well in these type of calculations[2,3].

Channel and Pipe flow theory

To solve flow in the configuration the two principles stated before need to be represented mathematically. The first principle can be represented by the continuity equation. In general, the fluid density ρ may vary in response to changes in fluid temperature and pressure. For a fixed control volume ∀ enclosed by a surface S a general statement of mass conservation is given by

$$\frac{\partial}{\partial t}\int_{\forall} \rho d\forall + \int_{S} \rho \vec{v} \cdot \vec{n}\, dS = 0 \qquad [1]$$

In which $\vec{v}$ is the velocity at a point and $\vec{n}$ is an outer unit normal vector to the surface S, and t is time. The first term represents the accumulation of mass in the control volume over time. For steady state flows the term

$$\frac{\partial}{\partial t}\int_{\forall} \rho d\forall = 0 \qquad [2]$$

At a surface point the dot product $\vec{v} \cdot \vec{n}$ gives the component of the velocity which crosses the surface. Hence the second term computes the net outflow of fluid across the entire surface of the

control volume. For steady compressible flow of fluid in a pipe the conservation of mass is referred to as the continuity principle as stated above and can be written as

$$\dot{m} = \int_A \rho\, v\, dA = \rho_1 V_1 A_1 = \rho_2 V_2 A_2 = \rho_1 Q_1 = \rho_2 Q_2 \quad [3]$$

in which $\dot{m}$ is the mass discharge through a pipe of cross sectional area A, and can be written as the product of the density ρ of the fluid with the mean velocity V of the fluid at any cross section of the pipe. In equation [3], Q represents the volumetric flow rate in the pipe.

The second principle that needs to be satisfied is the work-energy principle which states that the pressure drop between two junctions must be the same for all paths between the two junctions. Different relationships can be used to express the pressure drop as a function of volumetric flow rate Q. In general, the pressure drop flow rate relationship for any branch of elements of a junction node can be expressed as

$$\frac{\Delta p}{\rho} = h_f = \sum_{n=0}^{N} K_n Q^n \quad [4]$$

The general exponential formula equation [4] covers all pressure drop scenarios. The values of K_n and n in equation [4] are the variables dependent upon the pressure drop flow rate relationship used. N refers to the order of the equation used to describe the pressure drop.

Equation [4] is also valid for cases where pressure is added to a domain instead of being lost. When pressure is added to the domain it is a negative pressure loss and occurs when pumps add pressure to the system.

To solve the flow, the continuity equation [3] needs to be satisfied on each and every junction and equation [4] needs to satisfy the same pressure loss for all the paths between two junctions.

To solve the equation of the flow in the domain shown in figure 1, the flow needs to be represented on a computational domain.

Computational Domain

The flow domain shown in figure 1 needs to be broken into a number of elements and nodes for computational purposes. Figure 2 shows the computational domain presented in figure 1. Elements shown by their element numbers in ( ) brackets are connected by nodes shown by their node numbers in [ ] brackets. Each element joins two nodes and the connectivity arrows in figure 2 indicate the orientation of the element in the computation scheme. The element is characterized in the direction of the arrow. So as an example element (4) connects node [3] to node [4] and element (3) connects node [2] to node [5].

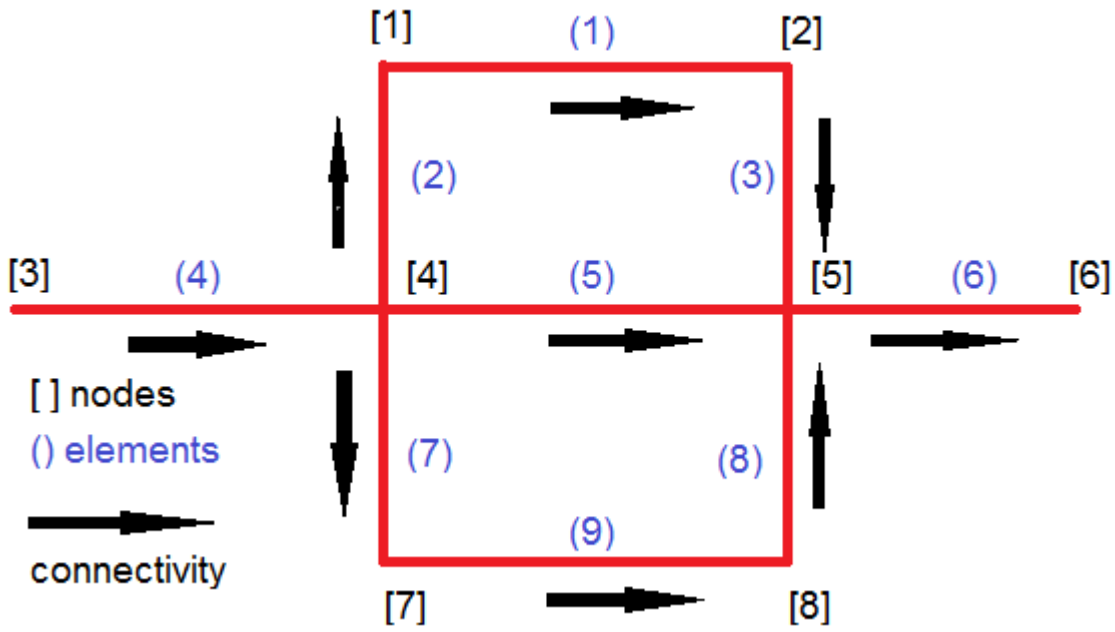


Figure 2. Computational domain of the system from in figure 1.

In figure 2 node [*i*] is connected through elements (e_{ij}) to neighboring nodes [n_{ij}] with J being the number of branches associated with node [*i*]. As an example, consider node [4], which has 4 branches associated with it. Node [4] is connected through elements (4), (2), (5) and (7) to nodes [3], [1], [5] and [7]. If the connectivity arrow is pointing toward the node the connectivity operator is (+) for the adjoining node. Hence for node [4], the only positive connectivity operator associated with it for is for node [3] which is connected to it through element (4). The other 3 connectivity operators are all (-).

Each node in the domain has a connectivity operator associated with each of its branches. This is vital in determining the direction of the flow in the branches or in the loops of the system.

Discretization

To solve equations [3] and [4] numerically, the equations must be represented algebraically on the computational domain. Equation [3], the continuity equation, needs to be satisfied on every node. Equation [4] needs to represent the pressure drop relationship in every element in the domain.

The continuity equation [3] can be expressed algebraically at each internal node by

$$\sum_{j=1}^{J} \rho_{ij} Q_{ij} C_j = \dot{m}_i \quad \text{i= 1, …Nodes, j=1..Branches} \qquad [5]$$

It must be noted that $\dot{m} = 0$ for all internal nodes and on the boundary nodes $\dot{m}$ equals the mass flux entering or leaving the domain on the boundary node. C_j refers to the connectivity operator for the branch of the node under consideration.

The work energy principle, equation [4], needs to be represented on the computational domain. In general the flow rate relationship for any of the branch elements (j) of node [i], figure 2, can be expressed as

$$\Delta p_{ij} = p_{nj} - p_i = C_{ij} \frac{\dot{m}_{ij}}{|\dot{m}|_{ij}} f(\rho_{i,j})\, h_{i,j} \qquad [6]$$

Where h_{ij} is the pressure drop flow rate relationship derived from equation [4], C_{ij} refers to the connectivity operator, $\dot{m}$ the mass flow rate, with f(ρ_{ij}) being a function of the density of the fluid in the domain. For incompressible flow f(ρ_{ij}) would be a constant however for compressible flows f(ρ_{ij}) would be a function of pressure and temperature and would need to be considered differently.

For an Ideal gas the relationship between density, pressure and temperature is given by the ideal gas law.

$$\rho_{ij} = \frac{p_{ij}}{RT_{ij}} \qquad [7]$$

where T_{ij} is the elemental temperature, p_{ij} the elemental pressure and R the gas constant. The elemental pressures are simply the mean of the two node pressures transforming equation [7] into

$$\rho_{ij} = \frac{(p_{ij} + p_{nj})}{2RT_{ij}} \quad [8]$$

The temperature in equation [8] can be obtained when solving the energy equation at the node [i]. The temperature of the fluid at node [i] can be obtained from the total energy of the fluid entering the node. The temperature of the fluid at node [i] is given by

$$T_{oi} = \frac{\sum_{j=1}^{J}(E_{ei,j} + \dot{m}_{ei,j}C_p T_{oi,j})_{inflow\ elements}}{\sum_{j=1}^{J}(\dot{m}_{ei,j}C_p)_{inflow\ elements}} \quad [9]$$

where T_o is the fluid temperature, $C_p$the specific heat of the fluid, $E_e$the energy transfer to the element e and $\dot{m}$ the mass flow rate in the element.

Equations [5], [6] and [9] are solved simultaneously till convergence is reached to obtain the flow rates, pressures and temperatures in the domain. In the case of incompressible flow, the density is constant and the link between the flow equations and the energy equation is very weak. The simultaneous solution of equations [5], [6] and [9] are required to satisfy the two principles of the solution. However, the next section will discuss in more detail the head loss relationships in the work energy equation [4] for channels or hoses, minor loss elements, and pumps.

Head Loss formulations

A form of the pressure loss term Δp or h_f given in equation [4] can represent the head loss due to fluid shear at the pipe or channel wall called pipe friction. Head loss due to pipe friction is always present throughout the length of the pipe or channel.

Additional head loss caused by local disruptions of the fluid stream can also be represented by another form of equation [4]. The local disruptions are also known as minor losses and are caused by valves, bends, tee junctions and other such fittings.

Pumps that add pressure to the system are negative head loss elements and can also be represented by equation [4].

Each of these head loss components represented by equation [4] will be considered individually.

Darcy-Weisbach formulation

The fundamentally most sound and versatile equation[4] for frictional head loss in a pipe is the Darcy-Weisbach equation and is given by

$$h_f = f\frac{L}{D}\frac{V^2}{2g} \quad [10]$$

where h_f is the pressure loss in meters, f the friction factor, V the mean fluid velocity in the pipe, D the diameter of the pipe and L the length of the pipe with g being gravitational acceleration. The friction factor f determines the frictional losses in the pipe and is dependent upon the equivalent sand grain roughness factor (e/D) of the pipe and the Reynolds number of the fluid flowing through it. The Reynolds number is given by equation [11]

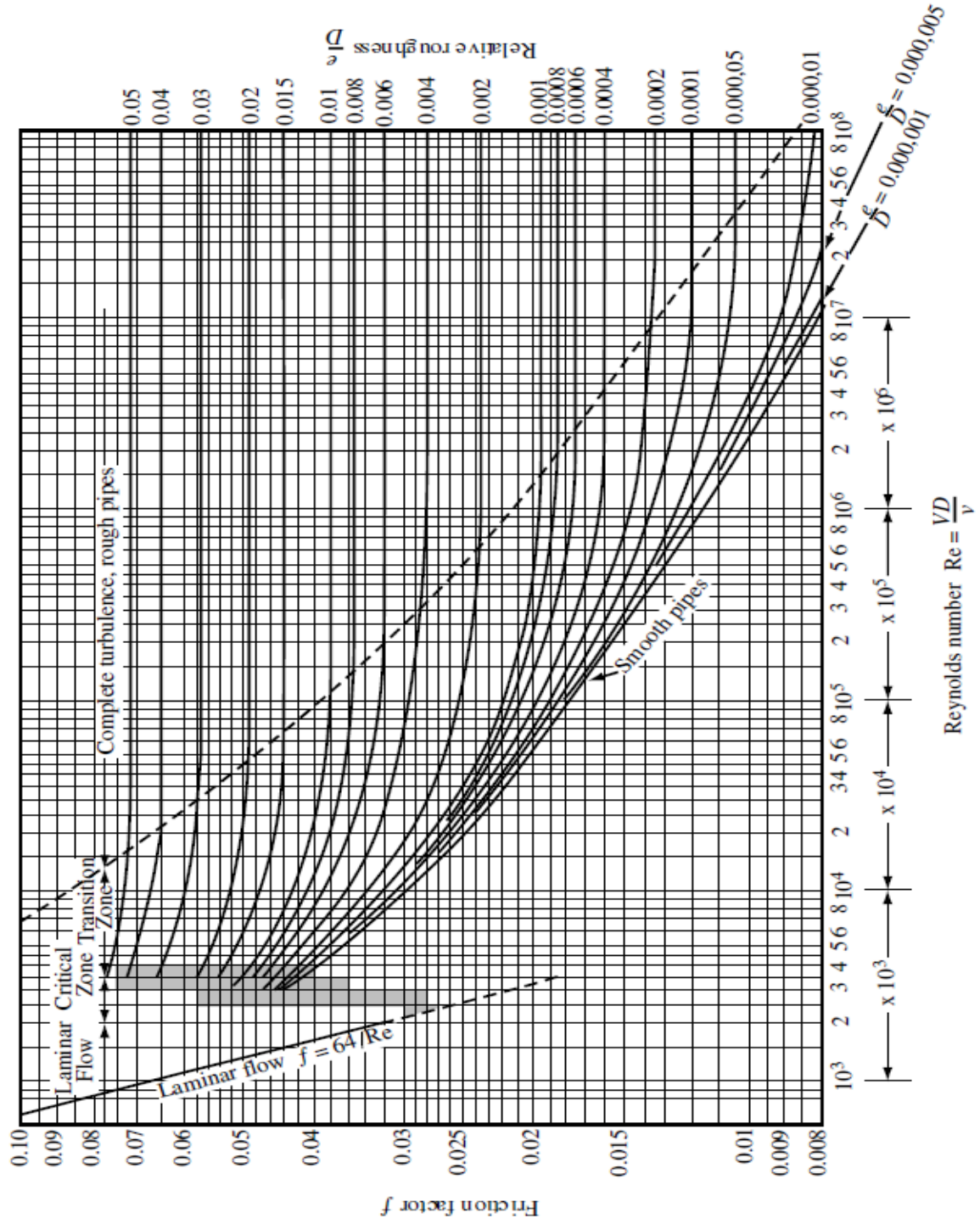
$$R_e = \frac{\rho V D}{\mu} \quad [11]$$

where μ is the viscosity of the fluid. For each pipe material either a single value or a range of sand grain roughness factors (e/D) values have been established. Table 1 presents common values for several commercial materials.

	e	
Material (new)	**ft**	**mm**
Glass	0.000001	0.0003
Drawn tubing	0.000005	0.0015
Steel, wrought iron	0.00015	0.046
Asphalted cast iron	0.0004	0.12
Galvanized iron	0.0005	0.15
Cast iron	0.00085	0.26
Wood stave	0.0006–0.003	0.18–0.9
Concrete	0.001–0.01	0.3–3.0
Riveted steel	0.003–0.03	0.9–9.0

Table 1 Average roughness of commercial materials. Reproduced from Mechanics of fluids Irving H. Shames[5].

The functional behavior of the dimensionless friction factor f is displayed fully in the Moody diagram figure 3.



From L. F. Moody, "Friction factors for Pipe Flow," *Trans.* A.S.M.E., Vol. 66, 1944

Figure 3 Moody diagram for the Darcy-Weisbach friction factor.

To solve the friction factor numerically the Moody diagram needs to be reduced to a form that can be programmed. There are several semi empirical correlation formulas relating the friction factor f to the Reynolds number Re to the relative roughness factor (e/D) for different Reynolds number ranges.

For Laminar flow with $Re < 2300$ the friction factor is given by

$$f = \frac{64}{Re} \quad [12]$$

as can be seen from the Moody diagram figure 3.

For fully turbulent flow where with $Re > 4000$ the friction factor is accurately represented by the Colebrook-White equation

$$\frac{1}{\sqrt{f}} = 1.14 - 2\log_{10}\left(\frac{e}{D} + \frac{9.35}{R_e\sqrt{f}}\right) \quad [13]$$

The Colebrook-White equation [13] is the original equation that combines experimental results of studies of turbulent flow in smooth and rough pipes to a common friction factor. Unfortunately, the equation is implicit in nature and needs to be solved iteratively. For this reason, many explicit approximations to the Colebrook-White equation exist along with other expanded forms. The Swamee-Jain equation is a very good approximation of the implicit Colebrook-White equation and is given by equation [14]

$$f = \frac{0.25}{\left[\log_{10}\left(\frac{e}{D} + \frac{5.74}{Re^{0.9}}\right)\right]^2} \quad [14]$$

For transitional flow i.e $2300 < Re < 4000$ a linear interpolation between equations [12] and its turbulent counterpart provides a good representative value of the friction factor. The most common favored approximation is the Swamee-Jain equation [14] and is the default approximation and used in this text. Other similar approximations that are commonly used are the Haaland equation

$$\frac{1}{\sqrt{f}} = -1.8\log\left[\left(\frac{\frac{\varepsilon}{D}}{3.7}\right)^{1.11} + \frac{6.9}{Re}\right] \quad [15]$$

Serghides approximation

$$\frac{1}{\sqrt{f}} = \Psi_1 - \frac{(\Psi_2 - \Psi_1)^2}{\Psi_2 - 2\Psi_1 + 4.781} \quad [16]$$

where

$$\Psi_1 = -2.0 \log\left[\left(\frac{\frac{\varepsilon}{D}}{3.7}\right) + \frac{12}{Re}\right] \quad [17]$$

$$\Psi_2 = -2.0 \log\left[\left(\frac{\frac{\varepsilon}{D}}{3.7}\right) + \frac{2.51\psi_1}{Re}\right] \quad [18]$$

$$\Psi_3 = -2.0 \log\left[\left(\frac{\frac{\varepsilon}{D}}{3.7}\right) + \frac{2.51\psi_2}{Re}\right] \quad [19]$$

Evangelides approximation

$$f = \frac{0.2479 - 0.0000947(7 - \log Re)^4}{\left(\log\left(\frac{\frac{\varepsilon}{D}}{3.615} + \frac{7.366}{Re^{0.9142}}\right)\right)^2} \quad [20]$$

and the Altshul Tsal equation

$$A = 0.11\left(\frac{68}{Re} + \varepsilon\right)^{0.25} \quad [21]$$

$$\text{If } A \geq 0.018\ f = A\ else\ f = 0.00258 + 0.85A \quad [22]$$

To correlate equation [10] to equation [4] the equation needs to be rewritten in terms of the volumetric flow rate.

$$h_f = f\frac{L}{D}\frac{Q^2}{2gA^2} \quad [23]$$

Then written in terms of pressure and not height

$$\Delta p = \rho f\frac{L}{D}\frac{Q^2}{2A^2} \quad [24]$$

Hence when compared to equation [4]

$$K_2 = \rho f\frac{L}{D}\frac{1}{2A^2} \quad [25]$$

Where $N = 2\ and\ K_0 = K_1 = K_3 = 0$

Once the relationship between the Darcy-Weisbach equation [10] and the work energy equation [4] has been established the final discretized form of the Darcy Weisbach equation can be established that corresponds to equation [6]. When comparing equation [24] to equation [6] it can be seen that

$$\Delta p_{ij} = p_{nj} - p_i = C_{ij}\frac{\dot{m}_{ij}}{|\dot{m}|_{ij}}\rho_{i,j} f_{i,j}\frac{L_{i,j}}{D_{i,j}}\frac{1}{2A_{i,j}^2}Q_{i,j}^2 \quad [26]$$

Equation [26] represents the final discretized form of the Darcy-Weisbach formulation.

Minor Losses

The Darcy-Weisbach equation calculates the pressure drop due to friction in a straight length of pipe. Minor losses are the pressure losses attributed to the flow flowing through fittings, valves, bends, elbows, tees, inlets, exits, enlargements and contractions. These losses are called minor losses in that, in theory, they should be minor compared to the friction losses. However, if the configuration has many bends, elbows with partially throttled valves, then these minor losses may be the cause of the largest pressure loss in the system. Flow through these valves and fittings is very complex and a theoretical analysis is not possible. Hence these minor losses are determined experimentally. Empirical relations are used for branching flows and these are widely published. Refer to Idelchick and Miller[6,7] for extensive studies on loss factors for components and branching flows.

Mathematically the minor losses are accounted for through the resistance coefficient or K factor. When the K factor for a component is readily available the pressure loss for the component can be represented as follows.

$$h_l = K\frac{V^2}{2g} = K_2\left(\frac{1}{2gA^2}\right)Q^2 \qquad [27]$$

By looking at the Darcy-Weisbach equation [10] the term $f\left(\frac{L}{D}\right)$ can be substituted with a loss coefficient K. From this correlation the minor loss can be expressed as an equivalent length of the existing pipe. Many texts refer to the minor losses in terms of equivalent lengths. Hence the discretized form of the Darcy-Weisbach equation [26] can be written to include minor losses.

$$\Delta p_{ij} = p_{nj} - p_i = C_{ij}\frac{\dot{m}_{ij}}{|\dot{m}|_{ij}}\rho_{i,j}\left(f_{i,j}\frac{L_{i,j}}{D_{i,j}} + K_{i,j}\right)\frac{1}{2A_{i,j}^2}Q_{i,j}^2 \qquad [28]$$

Equation [28] represents the discretized form of the pressure loss equation that takes pipe friction and minor losses into account.

The loss coefficient K depends upon the geometry of the component and the Reynolds number of the fluid flowing through it like the friction factor as seen by equation [28]. However, in most cases the K factor is assumed to be independent of the Reynolds number. This assumption is based on the fact that most flows in practice have large Reynolds numbers. The friction factor f and K

loss coefficient tend to be independent of the Reynolds number for flows with large Reynolds numbers. Refer to the Moody diagram figure 3.

Representative loss coefficients of K_2 for equation [27] are given in Table 2 for inlets, exits, bends, sudden and gradual contractions and valves. There is considerable uncertainty in these values since loss coefficients, in general, vary with the pipe diameter, surface roughness, Reynolds number and the geometry of the design. The loss coefficients of two identical valves from different manufacturers can vary significantly. The manufacturers data should be consulted for final designs rather than relying on generic information presented in handbooks. It is far safer and more representative to use actual manufacturer's empirical data for simulations rather than generic data.

Configurations also involve changes in direction without a change in diameter and such sections are called bends or elbows. The losses in these devices are due to flow separation on the inner side and the swirling secondary flows caused by different flow path lengths. The losses encountered during changes in direction can be minimized by making the turn easy on the fluid by using circular arcs instead of sharp turns. The generic loss coefficients for such bends are given in Table 2. Refer to Idelchick and Miller[6,7] for extensive studies on loss factors for components and branching flows.

General pressure losses

However if the flow is dependent upon the Reynolds number the pressure drop flow rate relationship needs to be taken into account for different flow rates. The pressure flow rate relationship can be represented by a polynomial and used in equation [6]. Cubic, parabolic or linear relationships can be represented by the following equation

$$\frac{\Delta p}{\rho} = h = \left(K_3 Q^3 + K_2 Q^2 + K_1 Q + K_0\right) \qquad [29]$$

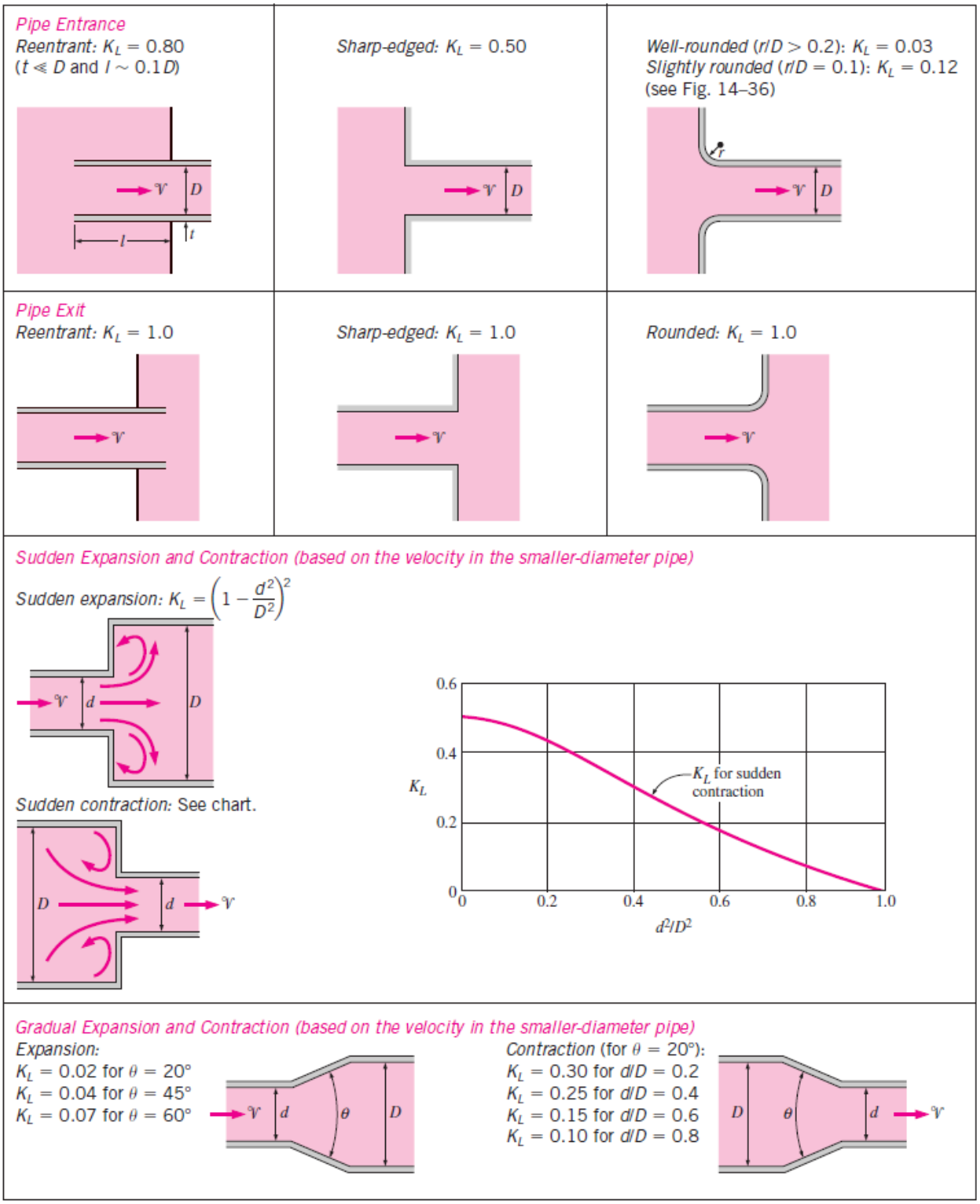
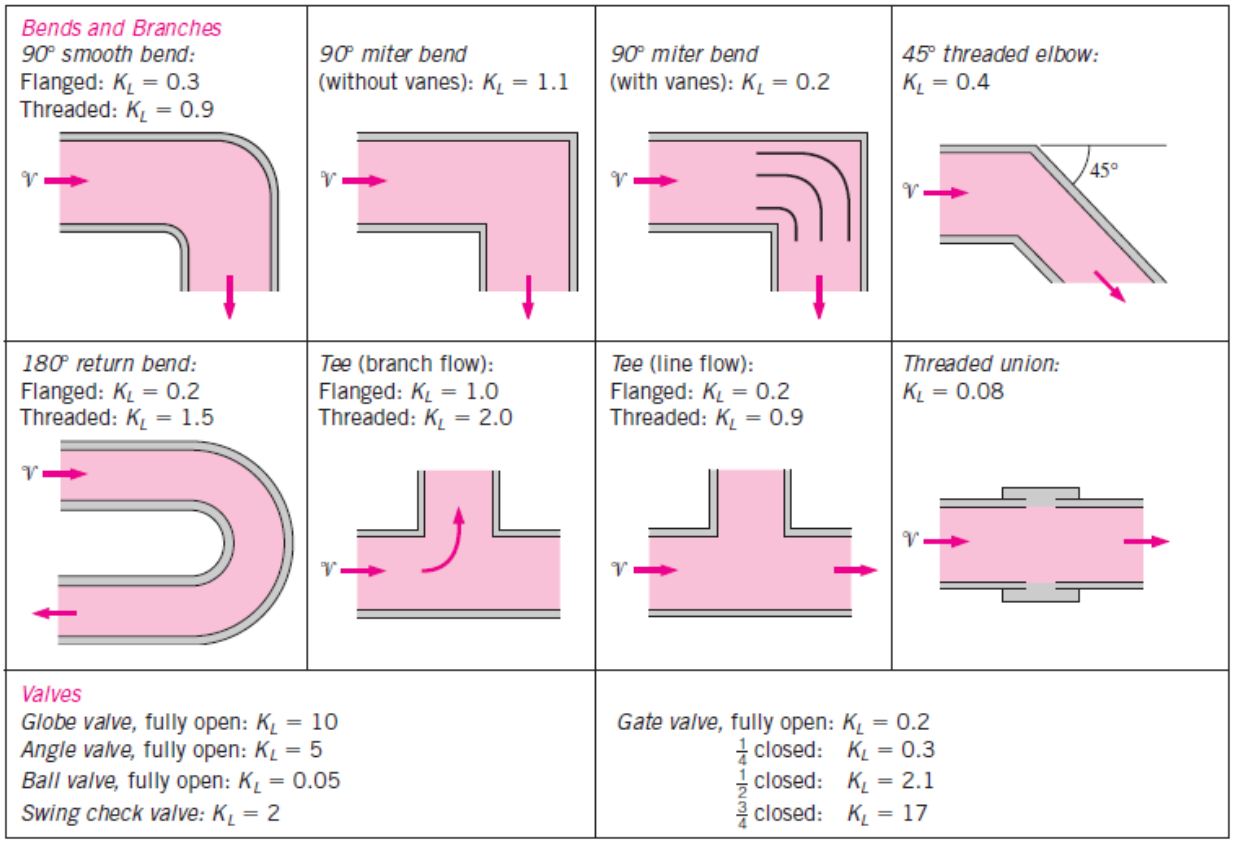


Table 2 Loss coefficients of various components for turbulent flow. (Reproduced from Fundamentals of Thermo-Fluid Sciences Yunus A. Cengel,John M. Cimbala, Robert H. Turner)



*These are representative values for loss coefficients. Actual values strongly depend on the design and manufacture of the components and may differ from the given values considerably (especially for valves). Actual manufacturer's data should be used in the final design.

Table 2 continued Loss coefficients of various components for turbulent flow. (Reproduced from Fundamentals of Thermo-Fluid Sciences Yunus A. Cengel,John M. Cimbala, Robert H. Turner[8])

Elevation

So far, the frictional losses causing pressure drops in fluid systems relating to pipes have been considered. The minor losses due to bends, valves and fittings have also been considered. To move a fluid through a configuration at a specified flow rate to a delivery station at an elevated point, the difference in elevation between the points needs to be considered. This is especially true if the fluid is an incompressible fluid with a high fluid density. If the origin of the configuration is at a lower elevation than the delivery point, a certain amount of positive pressure in the system is required to compensate for the elevation difference. However, if the delivery point is at a lower elevation than the origin, gravity would assist the flow, and the pressure required at the origin to move the fluid at a specified flow rate would be reduced by the elevation difference. Situations may exist where the elevation may have peaks and troughs that require careful consideration of the topology. In some

instances, the total pressure required to transport a fluid at a given flow rate through a configuration may depend more upon the ground elevation profile than the actual pressure drop due to frictional and minor losses. Generally, when considering the effects of elevation in a system the elevation of the inlets and outlets are of more importance than the actual peaks and troughs of the elevation within the system. Usually with no external energy added to a system the fluid will always flow from the point of higher elevation to the point of lower elevation irrespective of the highest point. The fluid will readily flow uphill in a system provided it is powered by the downhill fall to a lower elevation than the inlet. This is known as the siphoning effect and should be taken care of in the algorithm.

Pumps

Pumps are installed in cooling systems to provide the necessary pressure to compensate for frictional losses, minor losses and differences in elevation for dense fluids to provide the desired delivery pressure at a desired point in the cooling system. Pumps used in cooling systems are either positive displacement pumps or centrifugal pumps.

Positive displacement pumps generally have a higher efficiency with a fixed flow rate at any pressure. Positive displacement pumps include piston pumps, gear pumps and screw pumps. Centrifugal pumps are more flexible in terms of flow rates but have lower efficiencies and lower operating and maintenance costs. Hence most liquid systems are driven by centrifugal pumps.

Centrifugal pumps consist of one or more rotating impellers contained in a casing. The centrifugal force of rotation generates the pressure in the liquid as it goes from the suction side to the discharge side of the pump. Centrifugal pumps have a wide range of operating flow rates with good efficiency.

Since pumps or fans are designed to produce a pressure at a given flow rate the performance curve of a pump is a very important characteristic of the pump. The performance curve is a graphic representation of how the pressure generated by the pump or pump varies with the flow rate. Other parameters such as the efficiency, nett positive suction head and horsepower are also presented graphically as part of the pump performance curves. Figure 4 shows a general pump performance curve.

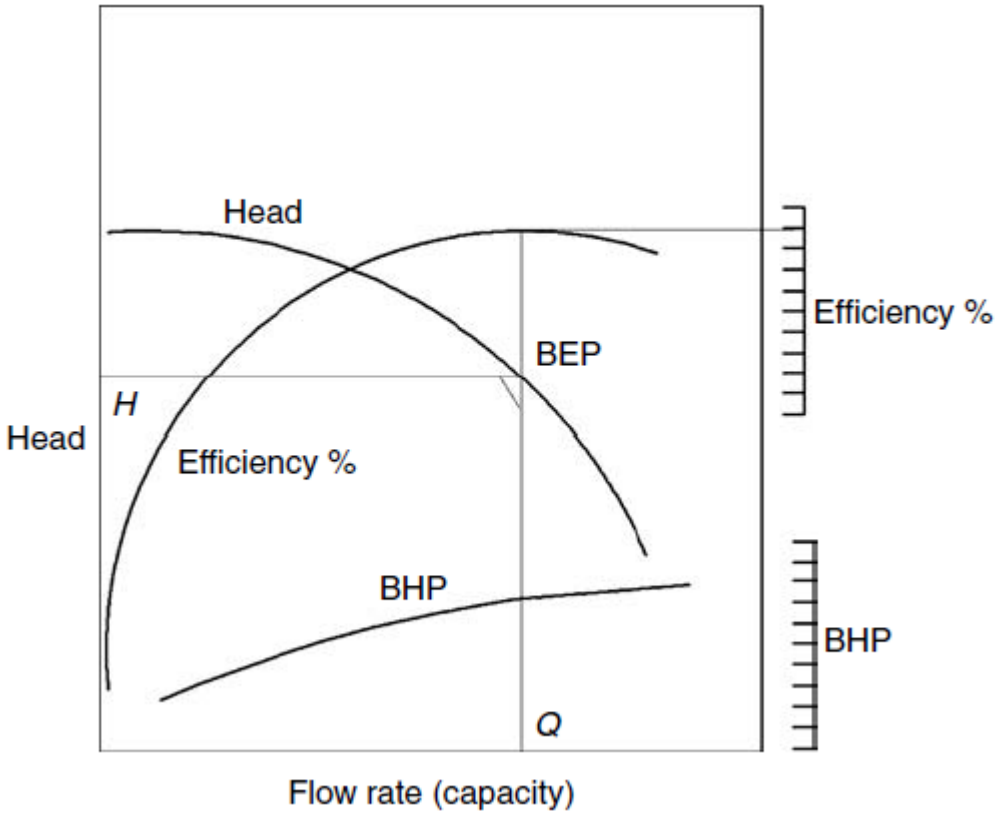


Figure 4 Generic performance curve for a centrifugal pump. (Image from Piping calculations manual, E. Shashi Menon[9])

The performance curves of a centrifugal pump consist of the head versus volumetric flow rate relationship, the efficiency versus volumetric flow rate relationship and the Brake Horse Power versus volumetric flow rate relationship. The term Head is used in preference to pressure when dealing with centrifugal pumps. The units of head are meters. Usually the fluid with the temperature used to generate the performance curve is provided with the curve. If the fluid is not provided then the reference density of the fluid should be provided. Using the reference density of the fluid, the Head versus flow rate curve can be written with the pressure specified in Pascal versus flow rate. Pump curves usually will have the reference fluid specified.

The head versus flow rate of the centrifugal pump characteristic curve figure 4 shows that the higher delivery pressures are generated at the lower flow rates. In general, the maximum head or pressure occurs at zero flow rate for pumps. This is known as the shut off head and the head decreases with an increase in flow rate

as shown in figure 4. The efficiency increases with flow rate up to the Best Efficiency Point (BEP) and then tapers off. The Brake Horse Power or (BHP) curve shows the horse power required to drive the pump for a flow rate on the Head versus flow rate curve.

The characteristic curve of a centrifugal pump depends upon the diameter of the pump impeller and the speed in revolutions per minute (RPM) at which the pump is operating. Pump manufacturers provide separate pump characteristic curves for each impeller size at each recommended RPM. Affinity laws for centrifugal pumps can be used to determine the pump performance at different speeds and minor variations in impeller diameters. However, it is recommended that exact curves be used instead of relying on affinity laws.

Another important pump characteristic that is provided by pump manufacturers when shipped is the net positive suction head curve also known as the (NPSH) of the pump. This represents the absolute minimum pressure allowed at the suction of the pump impeller for a specific flow rate that prevents pump cavitation. Pump cavitation occurs when the pressure in the suction casing of the pump falls below the vaporization pressure of the fluid and the fluid turns to vapor. This vapor is then carried over to the discharge side of the pump where the pressure is high and the vapor is compressed back to a liquid. This imploding action is violent in nature and has the potential to pit the surface of the impeller causing physical damage to the pump, prematurely shortening its operational life. Hence when designing systems, the available NPSH at the pump inlet always has to be more than the required NPSH specified on the pump vendor's performance data sheet.

Figure 5 is a pump curve from a pump manufacturer that is shipped with the pump. From the curve three head versus flow rate curves are given for three different impeller sizes. It can be seen from figure 5 that these curves are only valid for the pump operating at 1775RPM. It can also be seen that at zero flow rate the shut off head is the maximum for all 3 impeller sizes. Pump efficiency curves intersect the head curves and at approximately 450m^3/hr the pump is operating at its maximum efficiency which is approximately 86%. The NPSH curve is also provided on the same characteristic curve and if the designer can guarantee that the static pressure at the inlet is higher than 12m then there will never be any cavitation issues in the pump. The brake horsepower for the pump is also provided for each of the impeller sizes. It should also be noted that for this curve the reference density was not provided however it clearly states that the curve is based on pumping clear water below 85 degrees F. It should be noted that the characteristic curve

presented in figure 5 was reproduced from Hydraulics of Pipeline Systems by Bruce E. Larock, Roland W. Jeppson and Gary Z. Watters, CRC Press, page 518.

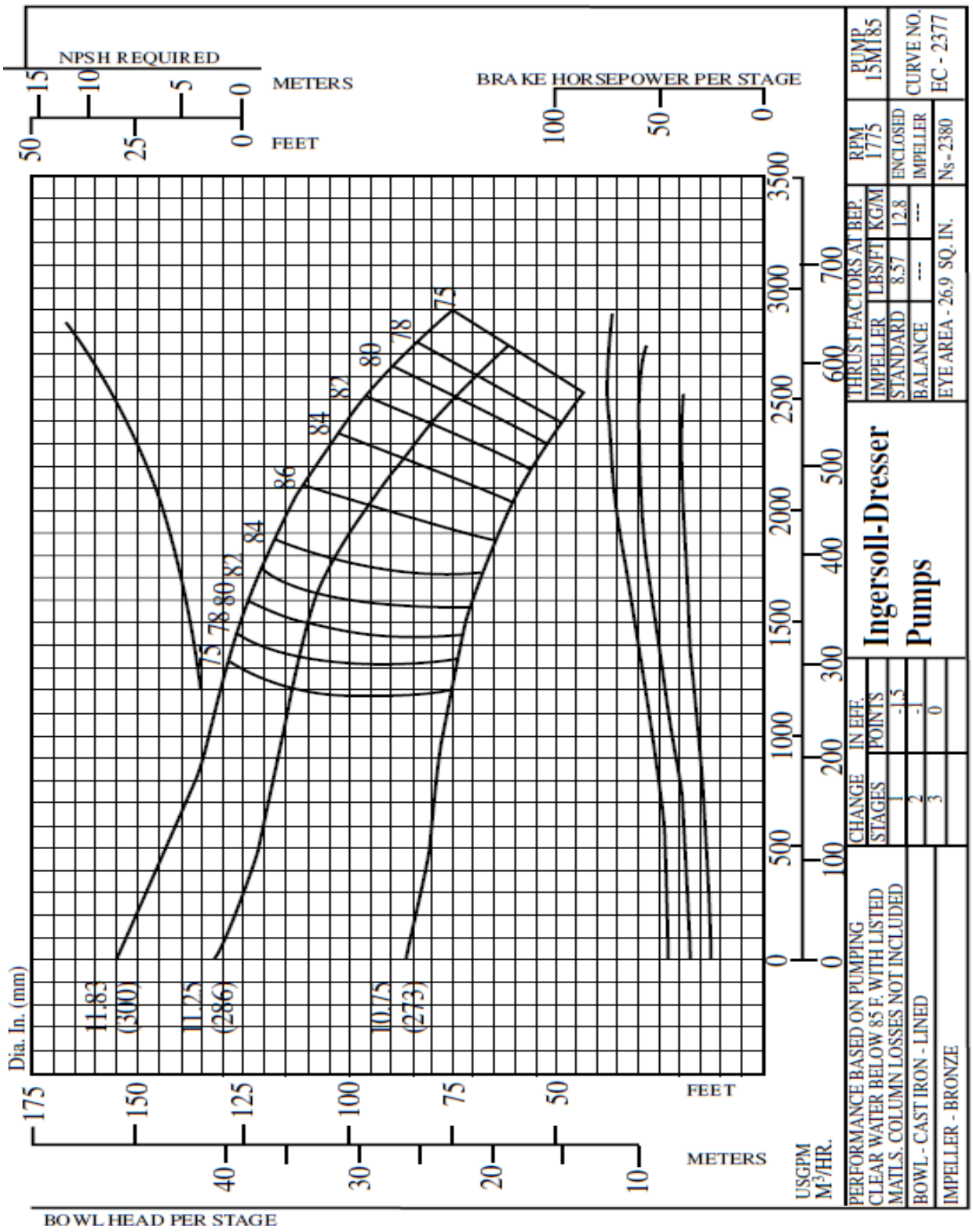


Figure 5 Pump characteristic curves provided by a pump manufacturer

To consider the pump curve for simulation purposes it has to be presented in a form that satisfies equation [4], the work energy equation. The head versus flow rate curve needs to be represented mathematically. The pump needs to be represented on the computational domain in the same form as the frictional and minor losses are in equation [26]. The pump needs to be modeled mathematically so that it can be represented in a form equivalent to equation [6]. To do this the characteristic curve needs to be expressed in the form of a polynomial equation like equation [29]. However, it should be noted that the pump provides pressure to the system and is modeled as a negative flow resistance; hence the sign needs to be reversed.

$$\frac{\Delta p}{\rho} = h = -\left(K_3\, Q^3 + K_2\, Q^2 + K_1 Q + K_0\right) \qquad [30]$$

It was noted previously, equation [6] that the flow rate relationship for any of the branch elements of node [i] given in figure 2 can be expressed by

$$\Delta p_{ij} = p_{nj} - p_i = C_{ij}\, \frac{\dot{m}_{ij}}{|\dot{m}|_{ij}} f(\rho_{i,j})\, h_{i,j} \qquad [6]$$

When expressing equation [30] in its final discretized form that conforms to equation [6] the pressure loss associated with a pump element is given by equation [31].

$$\Delta p_{ij} = p_{nj} - p_i = -C_{ij}\, \rho_{i,j} \left(\frac{\dot{m}_{ij}}{|\dot{m}|_{ij}} K_3 \left|Q_{i,j}^3\right| + \frac{\dot{m}_{ij}}{|\dot{m}|_{ij}} K_2 \left|Q_{i,j}^2\right| + \frac{\dot{m}_{ij}}{|\dot{m}|_{ij}} K_1 \left|Q_{i,j}\right| + K_0\right) \qquad [31]$$

Simulation implementation

To solve the flow in the system the correct pressures need to be computed on each node [ ] shown in figure 2 together with the correct flow rates in each element ( ). To do this the following equations need to be solved simultaneously. When presenting the following equations, it should be noted that the index i refers to the nodes of the domain and the index j to the branches of the elements attached to the node.

The continuity equation [5] needs to be satisfied at each node

$$\sum_{j=1}^{J} \rho_{ij} Q_{ij} C_j = \dot{m}_i \quad \text{i= 1, …Nodes, j=1,.. Branches} \qquad [5]$$

together with the flow rate pressure drop relationship at each branch connected to each node, equation [6]

$$\Delta p_{ij} = p_{nj} - p_i = C_{ij} \frac{\dot{m}_{ij}}{|\dot{m}|_{ij}} f(\rho_{i,j})\, h_{i,j}\ \mathrm{i} = 1, \ldots \mathrm{Nodes}, \mathrm{j} = 1, .. \mathrm{Branches} \quad [6]$$

Equation [6] is a generic equation that is applied to any pressure loss element. When pipe elements with minor losses form the branch a more specific relation of equation [6] is equation [28]

$$\Delta p_{ij} = p_{nj} - p_i = C_{ij} \frac{\dot{m}_{ij}}{|\dot{m}|_{ij}} \rho_{i,j} \left(f_{i,j} \frac{L_{i,j}}{D_{i,j}} + K_{i,j} \right) \frac{1}{2A_{i,j}^2} Q_{i,j}^2 \quad [28]$$

When the branch has a pump element attached to it, the relation that applies is equation [31]

$$\Delta p_{ij} = p_{nj} - p_i = -C_{ij}\, \rho_{i,j} \left(\frac{\dot{m}_{ij}}{|\dot{m}|_{ij}} K_3 \left|Q_{i,j}^3\right| + \frac{\dot{m}_{ij}}{|\dot{m}|_{ij}} K_2 \left|Q_{i,j}^2\right| + \frac{\dot{m}_{ij}}{|\dot{m}|_{ij}} K_1 \left|Q_{i,j}\right| + K_0 \right) \quad [31]$$

If the fluid being considered is compressible then the density of the fluid needs to be solved as well equation. [8]

$$\rho_{ij} = \frac{(p_{ij} + p_{nj})}{2RT_{ij}} \quad [8]$$

If the density is being calculated then the energy equation [9] also needs to be solved on node [i].

$$T_{oi} = \frac{\sum_{j=1}^{J} \left(E_{ei,j} + \dot{m}_{ei,j} C_p T_{oi,j}\right)_{inflow\ elements}}{\sum_{j=1}^{J} \left(\dot{m}_{ei,j} C_p\right)_{inflow\ elements}} \quad [9]$$

Equation [9] represents the temperatures of the fluid on the nodes in the domain. As noted previously the elevation is only considered on the boundary nodes.

Equations [5] and the various forms of equation [6] are solved simultaneously together with equations [8] and [9] to get the flow and temperature distribution in the domain. The solution scheme is based on a node method like that used in Computational Fluid Dynamics (CFD) to solver the Navier-Stokes equations. The scheme is based on the well-known SIMPLE algorithm of Patankar and Spalding[1], which is known in CFD as a segregated method. A similar method is detailed in Greyvenstein[2] and Greyvenstein and Laurie[3]. The matrices are solved using typical (CFD) solvers depending upon the matrix size, type and available hardware.

Boundary conditions

The simulation, or any simulation, is only as good as the boundary conditions provided. Accurate boundary conditions will provide accurate results. When looking at figure 2, the computational domain of a typical configuration, nodes [3] and [6] are the boundary nodes for this configuration. The definition of a boundary node is any node that is connected to one element. Boundary conditions need to be specified on all the boundary nodes in the domain.

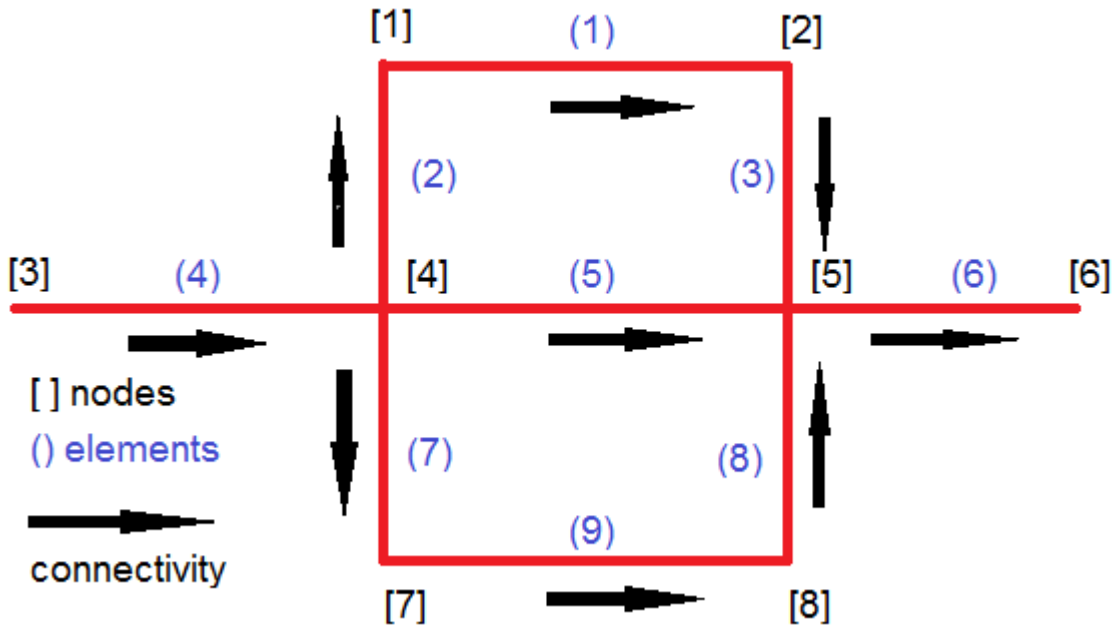


Figure 2 Computational domain of configuration in figure 1.

Flow boundary conditions

A flow boundary condition needs to be specified on all the boundary nodes in the domain. To satisfy the simultaneous solution of equations [5] and [6], either a pressure or a flow rate needs to be specified on each boundary node.

A pressure boundary condition is rather obvious and that is the precise pressure of the fluid at the boundary node. If fluid is discharging out to atmosphere, unrestricted, the typical pressure boundary condition would be atmospheric pressure or zero-gauge pressure.

From equation [3], the flow rate boundary condition can exist in several forms. If the density is known at the boundary node of an incompressible fluid, the boundary condition can be represented as a volumetric flow rate.

If the density is unknown the flow rate boundary condition can be represented as a mass flow rate.

From equation [3], the flow rate can also be represented as a velocity and by using equation [11], any flow rate can be represented by the Reynolds number. Hence a flow rate boundary condition can be specified as a volumetric flow rate, mass flow rate, velocity and Reynolds number.

When solving pipe systems, at least one boundary condition in the domain needs to be a pressure boundary condition. This is used to anchor the entire solution to a specific pressure.

Thermal boundary conditions

Equation [9] represents the energy equation of the fluid in the system. The energy equation requires accurate boundary conditions to get accurate temperature solutions in the domain. The boundary conditions for the energy equation only need to be specified on the inlet nodes. Hence the fluid temperature on the inlet node needs to be specified. By looking at equation [9] the term $E_{i,j}$ represents the external energy added to the system. This could be an external heat flux applied to the pipe modeled as a heat flux specified on the element. The energy could also be applied to the fluid in the pipe by specifying a fixed temperature to its surface. This can be modeled by specifying a fixed temperature to the element instead of the flux. The same mechanism for applying energy to the fluid is valid for pump. This energy term is not considered to be a boundary condition and is more a characteristic of the element in the branch. The only boundary condition required for the fluid is the fluid inlet temperature. The default for the $E_{i,j}$ term is zero flux meaning that the fluid in the element has no interaction with its surroundings. The fluid is perfectly insulated for the default case. For the default case the temperatures will stay constant throughout the system.

Elevation

As noted previously the elevation should be specified on each node in the domain. For incompressible fluids, liquids, the density is very high meaning that the elevation differences have a significant impact on the flow solution. As noted previously, due to the siphoning effect, the elevations are only important on the boundary nodes in the solution, although it is good practice to have the elevations specified on each node. The elevations can be calculated from the spatial nodal coordinates for each node provided the direction of the gravity vector is known.

Elemental Properties

Pipe or channel elements

By looking at equation [18] the discretized form of the Darcy-Weisbach equation for the pressure drop in pipe elements the diameter of the pipe is required, together with the cross-sectional area for the pressure drop calculation. For round elements the relationship is easily calculated, however if the element is not round the relationship may be different. Hence the wetted perimeter and cross-sectional area are required to calculate the hydraulic diameter of the element. The length of the element is required together with the friction factor. From equation [14] the friction factor depends upon the Reynolds number in the fluid and the relative roughness of the pipe material. The fluid properties need to be specified, density and viscosity, however these apply to the entire domain and are element independent. The relative roughness needs to be applied to the specific element and generic values can be obtained from Table 1.

From equation [26] it can also be seen that the loss coefficients K need to be specified as well. Generic values are available in Table 2. However, it should be noted that the solver can calculate generic K factor values for bends, elbows, tee junctions, changes in diameter and cross sectional area. These calculations are based on the work done by D.S Miller[7]. However, it would be far better for actual values to be specified when constructing the system and by so doing the solver would ignore the generic calculations and use those specified.

Pump elements

All pump manufacturers ship their products with a unique pump curve as shown in figure 5. This curve represents the pressure rise flow rate relationship through the pump. For the entire system the solver will calculate a flow resistance curve for all the pressure loss elements specified in the system. This flow resistance will then be connected to the pump curve and where the flow resistance plot intersects the pump curve will be the operating point of the pump. This will define the efficiency at which the pump is operating at. If the circuit and pipe work to the inlet and exit of the pump are modeled in exact detail then the solution from the solver at the inlet node will predict the precise suction head at the entrance to the pump. This pressure can be used to check whether the installation allows for sufficient NPSH. This is a very important parameter when designing pump installations.

However, the pump curve figure 5 needs to be transformed to a polynomial as shown in equation [30]. To do this at least 4 matching head versus flow rate points need to be entered to characterize equation [30]. The reference density of the fluid also needs to be specified to link the element to the rest of the system.

Pump installations in systems are very sensitive to the precise flow conditions and need to be selected very carefully. When selecting pumps when designing systems, it is advisable to first perform the simulation using conventional pressure and flow rate boundary conditions at the pump installation point. Once the desired flow rate and pressure at the pump installation point in the system has been arrived at, it is then advised that a pump be chosen which would provide this desired pressure at this desired flow rate at the pump's optimum efficiency. The pump curve should then be entered in the solver and checked. All the other "what if" scenarios involving the pump can be evaluated from there. Pump selection is very specific to the configuration and generic solutions to pump selection do not exist.

If a pump element provides energy to the fluid in the system then this can also be specified in terms of a flux or temperature, similar to pipe elements. The solver will then take this effect into account throughout the rest of the domain.

Coolant flow modelling in Moldflow.

The Coolant flow process is used within the existing Moldflow Synergy environment. An existing CAD model or study file can be imported or the user can start a new model from scratch, by modelling the channels as curves then meshing the curves to be one dimensional cooling channel elements. A user can also take and existing Autodesk Moldflow midplane, dual domain or 3D model and change the process to be Coolant flow. As shown in figure 6.

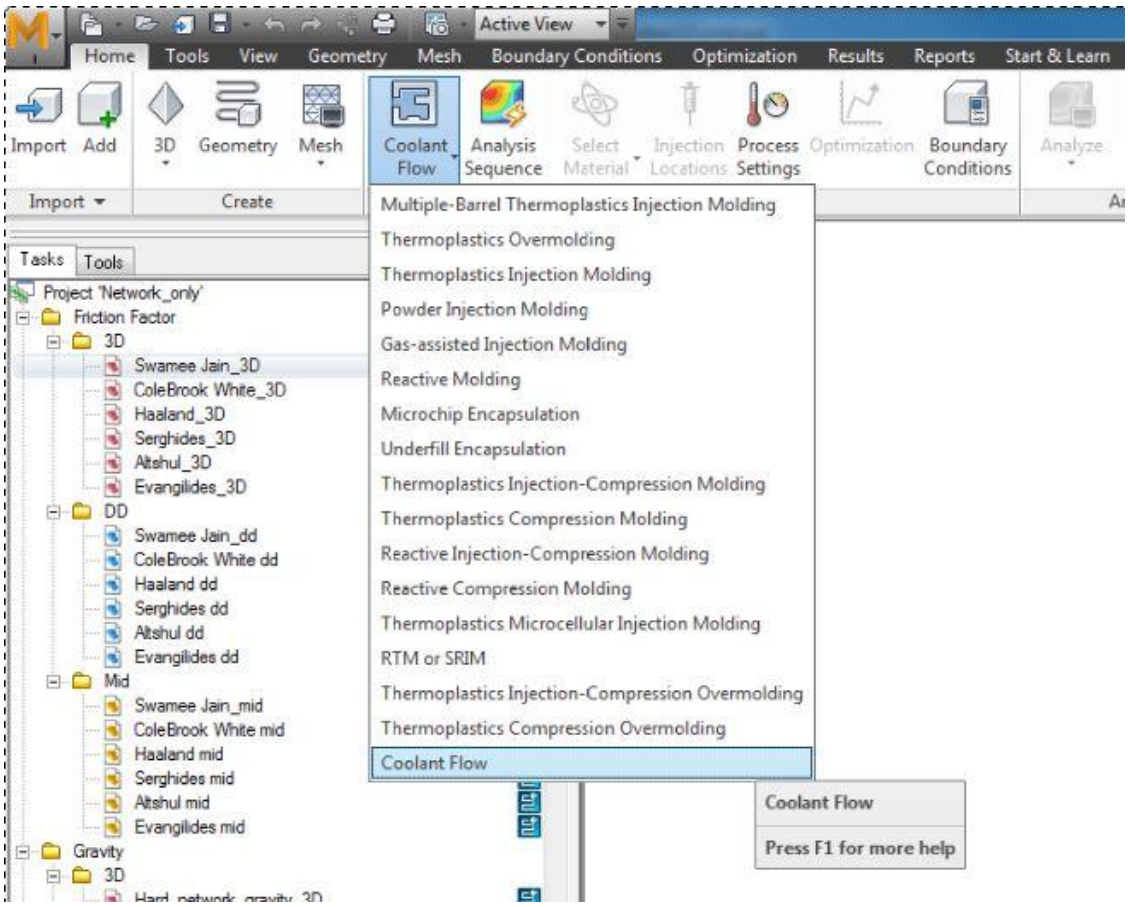


Figure 6 Process setting needs to be set to Coolant Flow

After the process has been set the user can model the cooling channels for analysis. Once the cooling channels have been modelled there are several new Process Settings options available related to the Coolant Flow process that are available for all Cooling analyses. These are shown in figure 7.

Process Settings

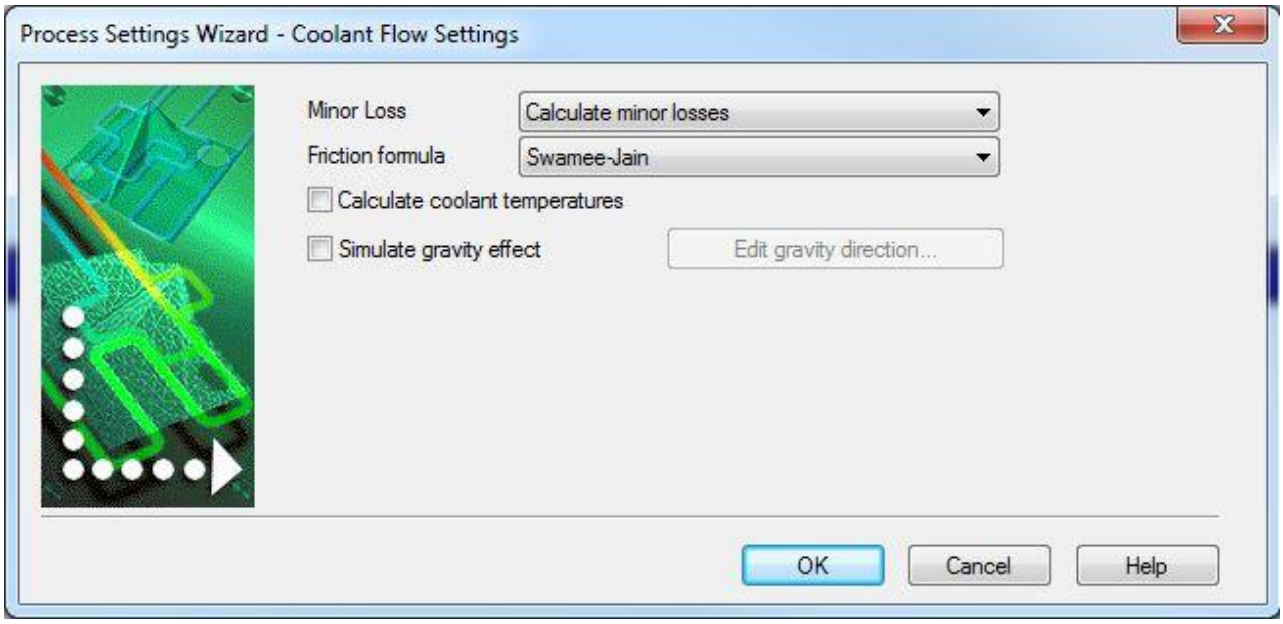


Figure 7 Process Settings for Coolant Flow analysis

When doing a Coolant Flow analysis, the user can select different minor loss options, options for specifying the friction formula loss definition, an option for solving the coolant temperatures when not doing a Moldflow cooling analysis and an option for taking gravitational effects into account.

Fluids moving through channels and pipes carry momentum and energy due to forces acting upon it such as pressure and gravity. Conversely forces acting upon the fluid can reduce its momentum and energy. Major force reducers are ***pipe friction losses*** and ***minor losses***.

Pipe friction losses are attributed to wall friction and shear losses that occur in the fluid between the wall of the channels and the fluid.

Minor losses are additional losses caused by bends, pipe entries and exits, fittings, elbows, valves etc.

The **Minor Loss option** allows the user 4 options when dealing with minor losses:

Calculate Minor Losses: This option is the default option that the Autodesk Moldflow Cool solvers use. The solver takes the geometry of the configuration into account and calculates a minor loss coefficient based on the angle of the bend of the channels. The solver also takes the changes in diameter, the entry and exit losses and branching flow into account. There are many different formulations for minor losses and the automatic calculation may not be accurate in all instances. When a more accurate value is required these other options can be used.

Use specified minor losses: When using this option, the user can specify minor loss coefficients on elements individually. The solver will use these instead of the automatic values when the automatic values are not trusted.

Calculate minor losses unless specified: This option will calculate the minor losses however if a user wants to override a specific minor loss value they just need to specify the loss on the element to override it. The solver will then calculate minor losses on elements except those that a value is specified on.

Ignore minor losses: With this option the minor losses are ignored and a minor loss of 0.0 is used on each element.

It is noted that the Coolant Flow solver does output the minor loss coefficient as a result and the existing Cool solvers have been enhanced to have the same results as the Coolant Flow solver. All existing Autodesk Moldflow Cool analyses have the same options as the Coolant Flow solver.

The **Friction formula -Option** allows for the selection of the friction loss approximation. Frictional losses in both laminar and turbulent flows are obtained from the Moody diagram Figure 3. The Moody diagram is a chart in non-dimensional form that relates the Darcy Weisbach friction factor, Reynolds number and relative channel roughness for fully developed flow in a circular pipe equation [10]. The Moody diagram, figure 3, is based on experimental results.

With the Coolant Flow solver, the Friction formula option allows the user to choose different mathematical representations of the Moody diagram. It is noted that the Colebrook White formulation, equation [13], is the exact solution for the Moody diagram. However, the Colebrook white formula is explicit in $\sqrt{f}$ where f is the friction factor. $\sqrt{f}$ appears on both the left and right sides of the equation and can only be solved in an iterative manner. Solving this equation can be time consuming hence all the other formulations are implicit approximations to the Colebrook White formulation. They are much easier to solve than the Colebrook White equation. The solver does support the Colebrook White equation; however the analysis time of the overall solution will be much slower. Autodesk Moldflow Cool solver uses the Swamee Jain equation, equation as default.

All supported options are

- ***Swamee Jain* (default),** equation [14]
- ***Colebrook White*,** equation [13**]**
- ***Haaland*,** equation [15]
- ***Serghides*,** equations [16] to [19]
- ***Altshul*** equations [21] to [22]
- ***Evangelides*** equation [20]

There are numerous texts and theses detailing the validity of each formulation for a flow regime.

The **Calculate coolant temperatures** allows the user to solve the temperature of the fluid in the system when checked. Equation [9]. However, when doing a Coolant Flow analysis, the part and mold is not included in the simulation. Heat transfer options can be specified on an element as a heat transfer boundary condition that will influence the temperature of the fluid in the channels, figure 9. The fluid inlet temperature entering the system needs to be specified on the inlet node when using this option. Then depending on the specified boundary conditions of the elements in the system the effect of these boundary conditions on the temperature of the fluid inside the configuration will be calculated. When doing a conventional cool analysis, the mold temperature solution in the mold provides this boundary condition, hence this option is not available for conventional cool analyses.

When moving dense fluids, such as water, through channels that vary in height, the weight of the fluid begins to influence the energy required to move the fluid at a certain flow rate. Hence the

Simulate gravity effect, option is there to simulate the effect of gravity in the system. When doing a Moldflow Flow analysis, the parting plane of the mold is specified to be on the X-Y plane. The clamping force required is calculated using the projected cross-sectional area of the part on this plane. Hence the default gravity direction should be in the negative y direction. If the user intends to use gravity in a Moldflow analysis, it is advisable that the cooling channels are modelled consistent with the existing Moldflow conventions. This option is now available for all the Cooling analyses.

Element options

With all channel elements for a Coolant Flow or Cool analyses, the user can now specify the Loss Coefficient K factor or minor loss on the element, figure 8. This ties in with the Minor loss option on the Process Settings dialog box, figure 7, of the Coolant Flow analysis or the Advanced options dialog box for all existing Cool analyses. Based on the selection in the Process settings, the solver will either use the value specified in the Loss Coefficient K factor box or not.

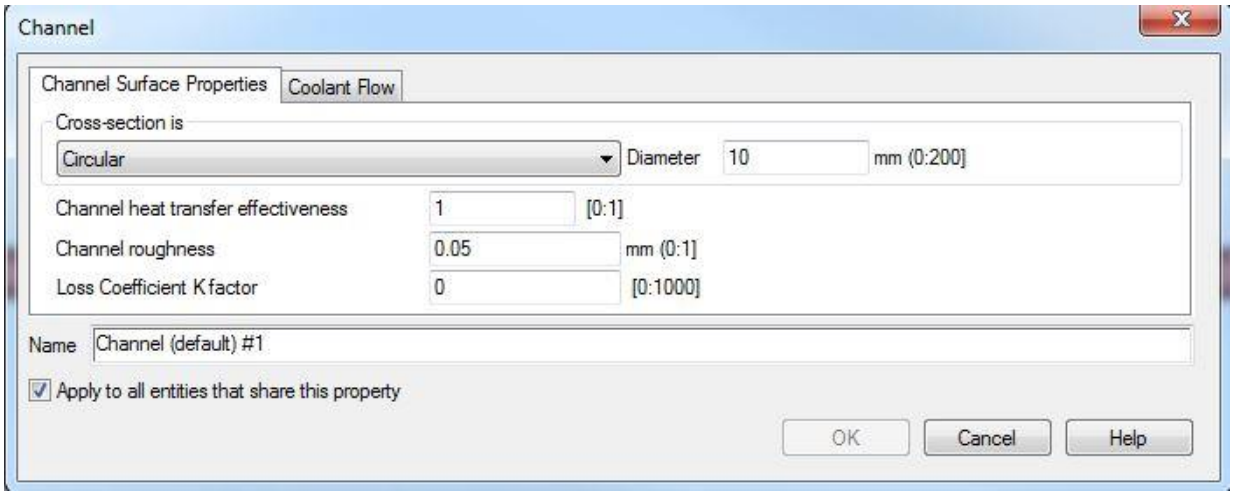


Figure 8 Channel properties showing Loss Coefficient Kfactor

The new Coolant Flow tab on the channel properties box, figure 9, is only available if the Coolant Flow analysis type is chosen, figure 6. This allows the user to specify heat transfer boundary conditions for the Coolant Flow analysis that are ignored if a Cool analysis is performed. These values are only used if the Calculate coolant temperatures checkbox is checked on the Process Settings of the Coolant Flow analysis, figure 7.

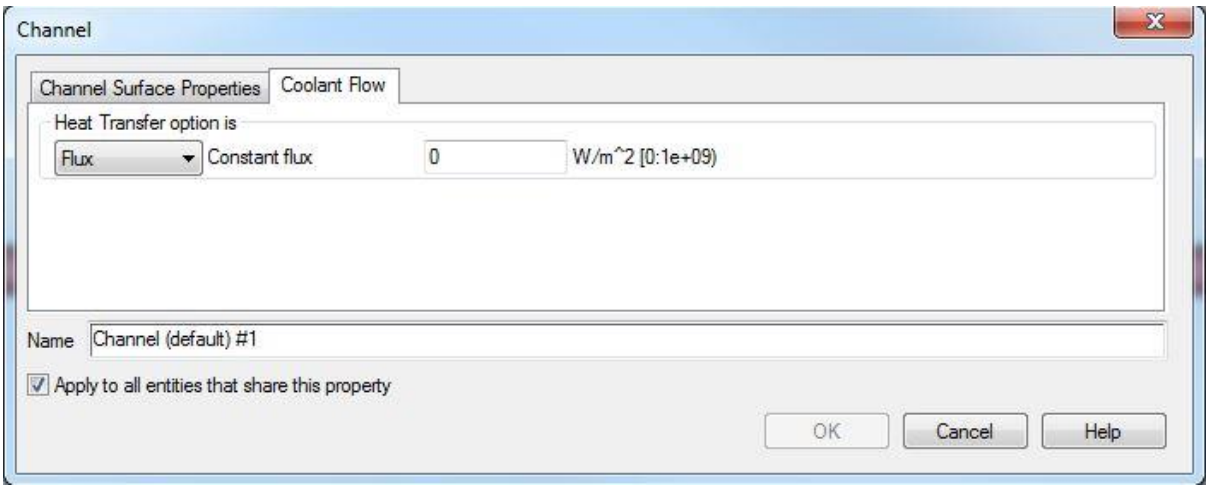


Figure 9 Coolant Flow tab on the channel dialog box

There is also a Heat transfer option section on the Coolant Flow properties box that relates to the coolant temperature equation, equation [9] and ties in with the “Solve coolant temperatures” option discussed previously. This section allows for 3 heat transfer options on the element.

Flux: The user can specify the heat flux on the channel element in Watts per meter squared. The convention is that a positive value shows the heat entering the circuit through the entire surface area of the element.

Total Heat: The total heat entering the element can be specified in Watts. If the user does not know the surface area of the element or only knows that a certain amount of heat is applied to the system in a certain area this option can be used. The solver will calculate the surface area of the element and convert the total heat into a flux and calculate its effect on the temperature of the fluid. Same convention applies as for the Flux option.

Temperature: The fixed temperature of the wall of the channel can be specified in (°C). If the surface of the element is maintained at a constant temperature the solver will calculate the heat transfer coefficient between the surface and the fluid, and the heat flux on the surface area of the element.

It is noted that these heat transfer options are ignored when doing Moldflow cooling analyses as the temperature solution in mold the provides the Coolant solver these actual temperatures.

Pump elements

A new element type has been introduced for the Coolant Flow analysis and this is the pump element shown in figure 10.

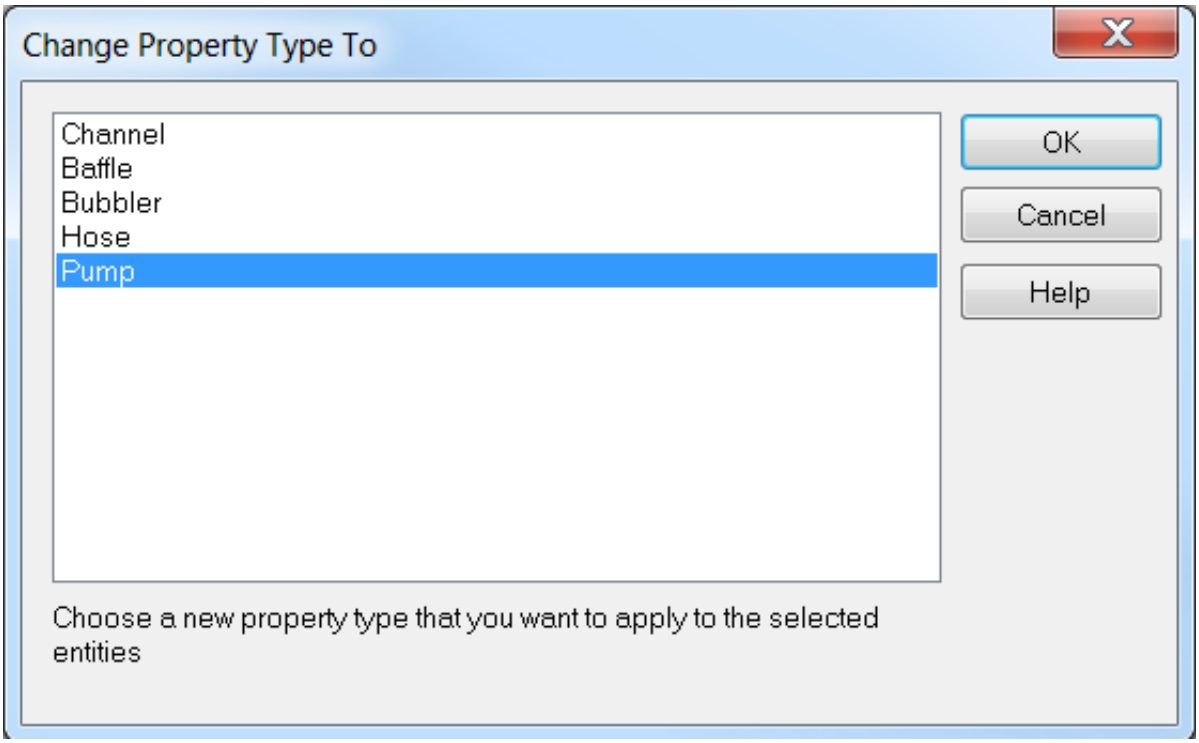


Figure 10 New element type Pump.

The element type, pump, figure 11, allows the user to specify the pump curve of the pump that puts the energy into the fluid to allow the fluid to flow through the system. The solver will calculate the operating point of the system, where the system resistance curve intersects the pumping pressure flow rate curve. When this option is used, it is important to model the hoses and exact geometry from the pump to the relevant circuit in the mold as all these components have a significant effect on the resistance of the cooling line. It is also important to simulate gravity too when using this option as the pump is always on a different level to the mold. Figure 11 shows the elemental dialog box for the pump element.

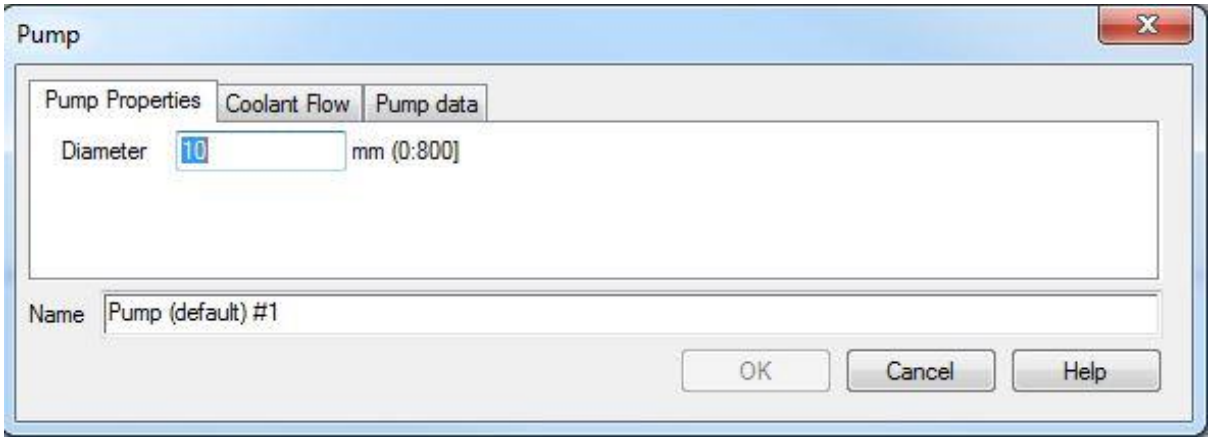


Figure 11 Pump properties dialog box

From figure 11 a diameter of the pump needs to be specified. This is more for display purposes in Synergy and is not used in the solver. The Coolant Flow tab is the same as that in the Channel properties dialog box, figure 9. The important tab for the pump is the Pump data tab as shown in figure 12.

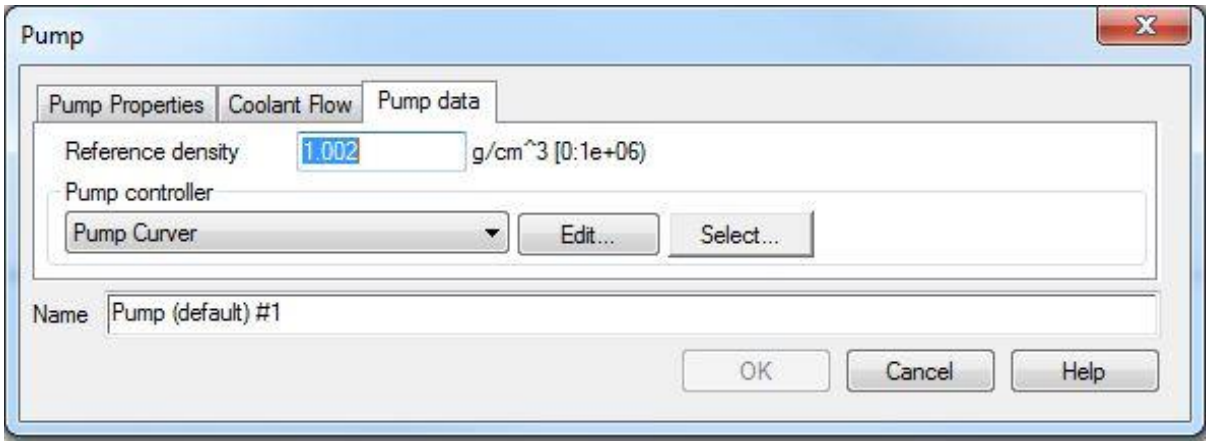


Figure 12 Pump data dialog box

The pump data dialog box shown in figure 12 allows the user to enter the pump curve. The first value that the user needs to enter is the reference density. The reference density is always specified on the pump manufacturers curve figure 5. It is the density of the fluid that was used to create the pump curve. This value needs to be entered together with the pump curve that is selected in the pump controller. Various pump curves can be selected figure 13.

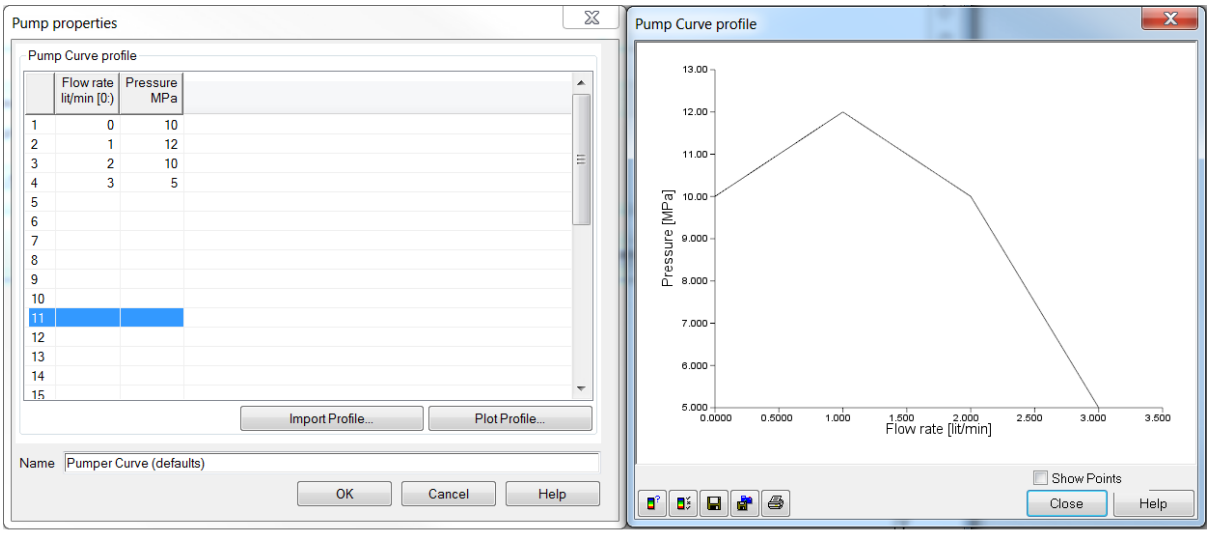


Figure 13 Pump curve specification

When entering a pump curve at least 4 points from the pressure versus flowrate from the manufacturer's pump curve, figure 5, needs to be entered as shown in figure 13. It is preferable that many more than 4 data points be used for more accurate results. Best results are achieved when more than 20 data points are used. The solver will then calculate the operating point on the curve where the circuit resistance intersects the pump curve. All pumps come with an efficiency curve, figure 5, and from that curve the efficiency of the pump can be deduced as well. What is very important when using this option is that all the hoses and plumbing be modelled accurately as this has a big effect on the accuracy of the solution.

It is also important when using pumps in the system to use a pressure boundary condition on the inlet node. A flow rate or Reynolds number boundary condition can be supported however the solver will force that flowrate through the pump and the pressure rise at that flow rate will be fixed from the pump curve. This is not wrong, as the pump will need to be throttled if it is powerful enough to achieve the flow rate. If this is done there may be no point in using a pump curve in the analyses when using flow boundary conditions.

Coolant Flow result type

All the coolant flow results that are available for an Autodesk Moldflow cooling analysis are available for a Coolant Flow analysis and vice versa. This means two new results are available for existing Moldflow Cool analyses.

The new results are:

Circuit loss coefficient or K factor: As discussed previously the minor loss coefficient commonly known as the K factor is given as a result. This allows users to check the calculated value and to also make sure that the specified values are used as expected.

Friction factor: When using different friction factor formulations that are specified in the process settings, figure 7, the user can verify that the actual friction factor matches the Moody chart, figure 3.

The existing cooling circuit results still available are:

- Circuit coolant temperature
- Circuit flow rate
- Circuit Reynolds number
- Circuit metal temperature
- Circuit pressure
- Circuit heat removal efficiency
- Circuit heat flux

Users can refer to the existing cool solver documentation to understand the interpretation of these results.

Results

To demonstrate the results from the different friction factor formulation results, equations [13] to [22], table 3 shows the different friction factors and pressure drops derived from a 1m straight channel of 10mm in diameter with water flowing through at a Reynolds number of 10000. The result from the Colebrook white equation, equation [13] is the actual representation of the Moody diagram figure [3]

From table 3, all the friction factors are very similar to the Colebrook white equation and this proves that all the equations have been implemented correctly.

~~Implemetation~~Implementation	Friction Factor	Pressure drop (kPa)
Swamee Jain (default)	0.0383	2.008
Colebrook White	0.0376	1.980
Haaland equation	0.0374	1.972
Serghides	0.0376	1.980
Altshul	0.0363	1.926
Evangelides	0.0378	1.986

Table 3 Friction formula comparison for turbulent flow

Figure 14 shows the typical hose structure around an injection molding machine showing the coolant channels in the mold with the relevant plumbing around the injection mold. The cooling system consists of a pump at the base of the machine feeding coolant to injection mold. The cooling channel then exits the mold and flows back towards the pump and exits at a very high point, representative of the inlet to a cooling tower. The coolant will be cooled whilst falling inside the cooling tower and will be pumped through the mold again from the base. This test is used to demonstrate the usage of a pump in the simulation and the effect of elevation on the system resistance.

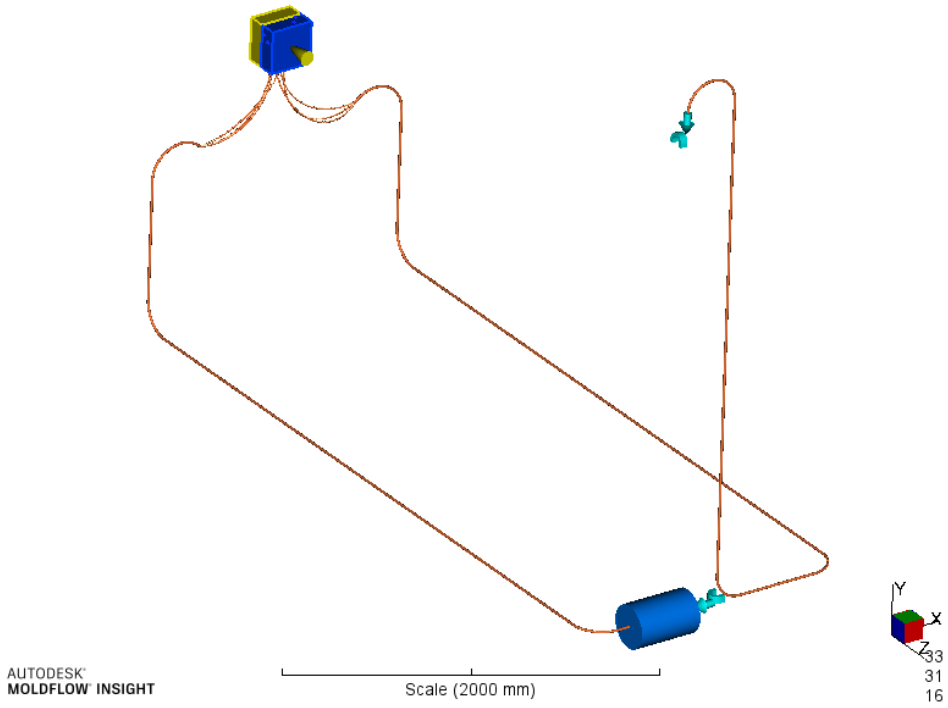


Figure 14 Cooling system for injection mold including pump

When the gravity is ignored and the check box on the process settings figure 7 is unchecked the flowrate through the cooling channel is 9.72 liter/min as shown on the blue demarcated line on the pump curve figure 15. Figure 15 shows that for this flow rate the pump will be adding approximately 20 kPa pressure to the fluid in the system.

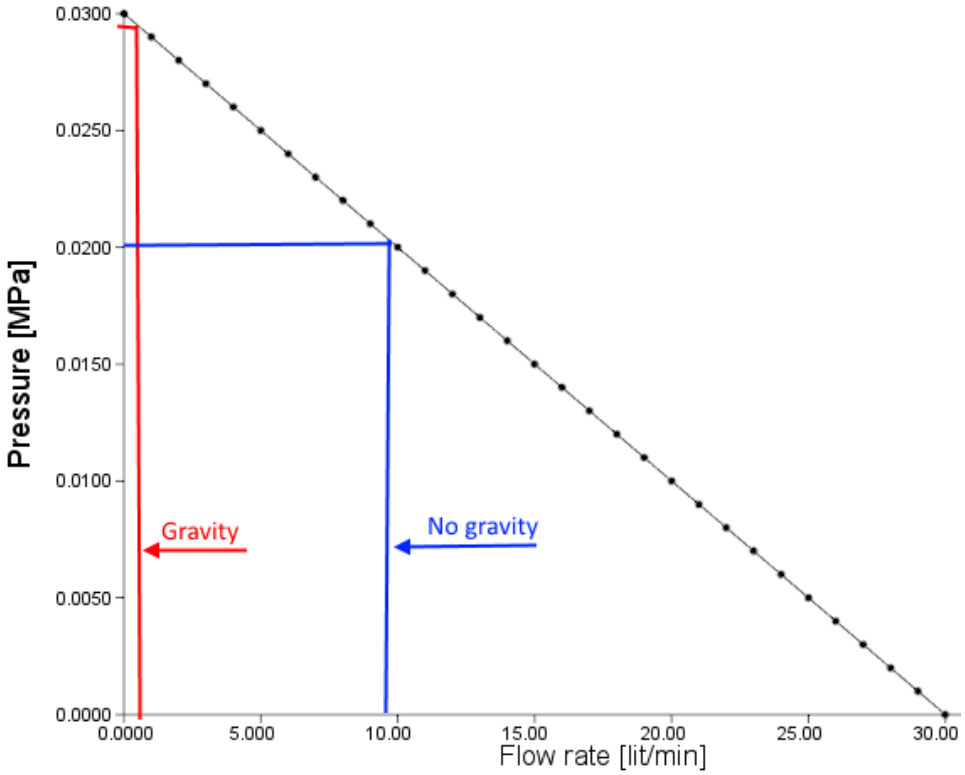


Figure 15 Pump curve operating points with and without gravity

By looking at figure 16, the difference between maximum pressure 19.79kPa and the minimum pressure -0.112 kPa in the system equals 19.902 KPa. This maximum pressure less the minimum pressure is the pressure difference across the pump. This 19.902 kPa and matches the blue “No gravity” operating point shown on the pump curve figure 15. The conclusion is that the pump curve simulation is working as expected as the flow rate and pressure increase from the results matches that on the pump curve.

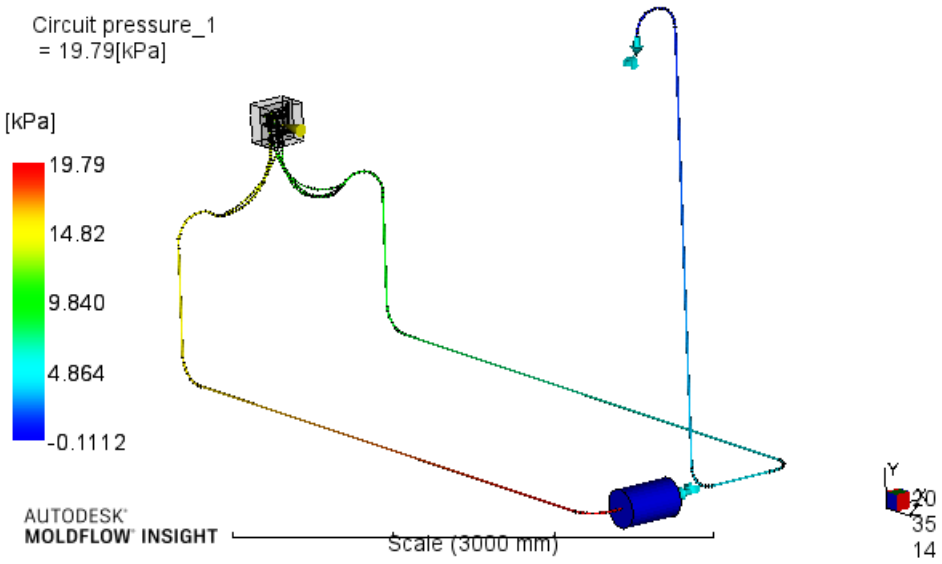


Figure 16 Pressure distribution in the system ignoring gravity

If gravity is included, by checking the Simulate gravity effect check box, figure 7, the flowrate through the system decreases from 9.72 l/min to 0.47 l/min. The difference between the maximum and minimum pressures in the system becomes 29.18kPa, shown in figure 17. This 29.18 kPa corresponds to the pressure rise over the pump shown on the red “Gravity” demarcation on the pump curve figure 15. Hence the pump curve simulation is working as expected when gravity is included in the simulation.

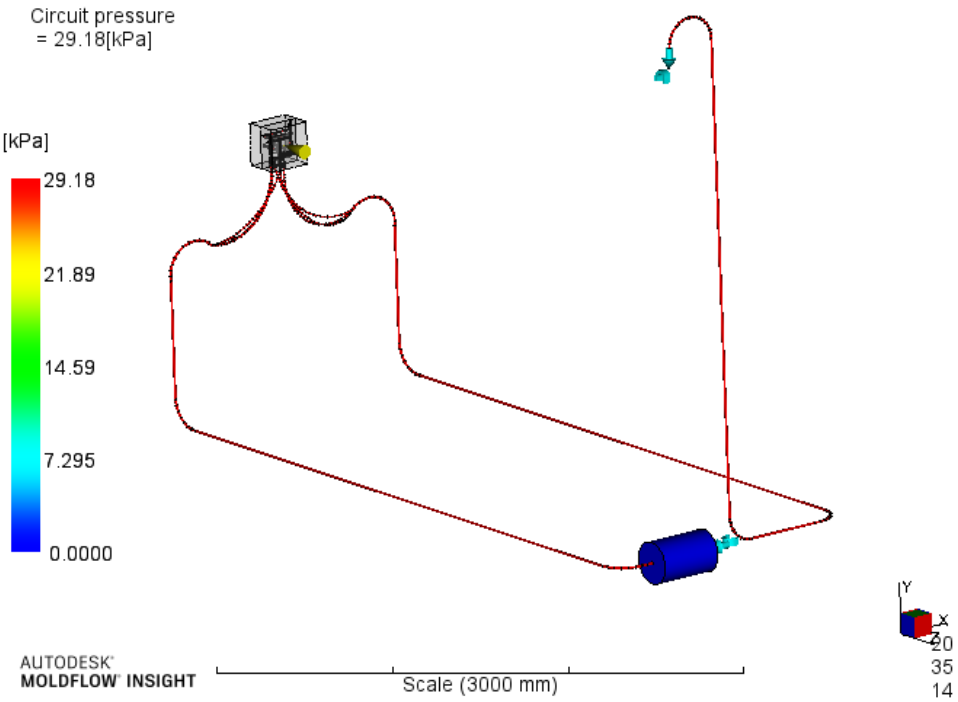


Figure 17 Pressure distribution in the system considering gravity

This exercise proves that if gravity is included, the pressure required to drive the fluid through the system is much higher. This pump can deliver the higher required pressure; however it is at a much lower flowrate. The flow rate decreases so much that the flow is laminar through the mold which is very bad for cooling and efficient heat transfer.

	No Gravity	Gravity
Flow rate (l/min)	9.72	0.47
Pressure difference (kPa)	19.902	29.18
Reynolds number max	15298.4	746
Coolant temperature rise (°C)	0.7	8.9
Heat removal (kW)	0.442	0.29
Cavity temperature (°C)	41.3424	68.3029
Mold temperature (°C)	29.3536	53.4187

Table 4 Gravity effect on mold results

Table 4 compares the Moldflow Cool analysis results on the mold for the case where gravity is included and excluded. As can be seen from table 4, if gravity is included the maximum Reynolds

number decreases from 15298.4 to 746. The flow goes from fully turbulent to laminar which results in poor mold cooling. The temperature rise across the cooling channel increases from 0.7 °C to 8.9 °C when gravity is considered. Even though the temperature rise across the channel increases, the heat removal decreases from 0.442 kW to 0.29 kW, due to the lower flowrate of the coolant, 0.47 l/min as opposed to 9.72 l/min. The lower heat removal results in the average temperature of the mold in contact with the part increasing from 41.3424 °C to 68.3029°C. That is a big increase in mold temperature and shows how inefficient the cooling system is when using the unthrottled pump, figure 15, with a cooling tower. The average mold surface temperature also increases from 29.3536°C to 53.4187 °C.

From this demonstration it can be shown that if gravity is ignored in this typical molding shop scenario, with the selected unthrottled pump, the impact on the accuracy of the solution can be detrimental.

Figure 18 shows the Circuit minor loss coefficient that is calculated by the solver for various 90 ° bends and “T” section scenarios shown in the figure. From figure 18 it can be see that the minor loss K factor is 1.012 for a 90 ° bend and for the “T” sections approximately 1.8. The published values in table 2 show that the 90° bend is 1.1 and the “T” sections approximately 2.0. From these results, the calculated results are a very good approximation to the published values, however not the same. Hence the user has the option for specifying minor loss coefficients in the solver if the calculated values are not accurate enough.

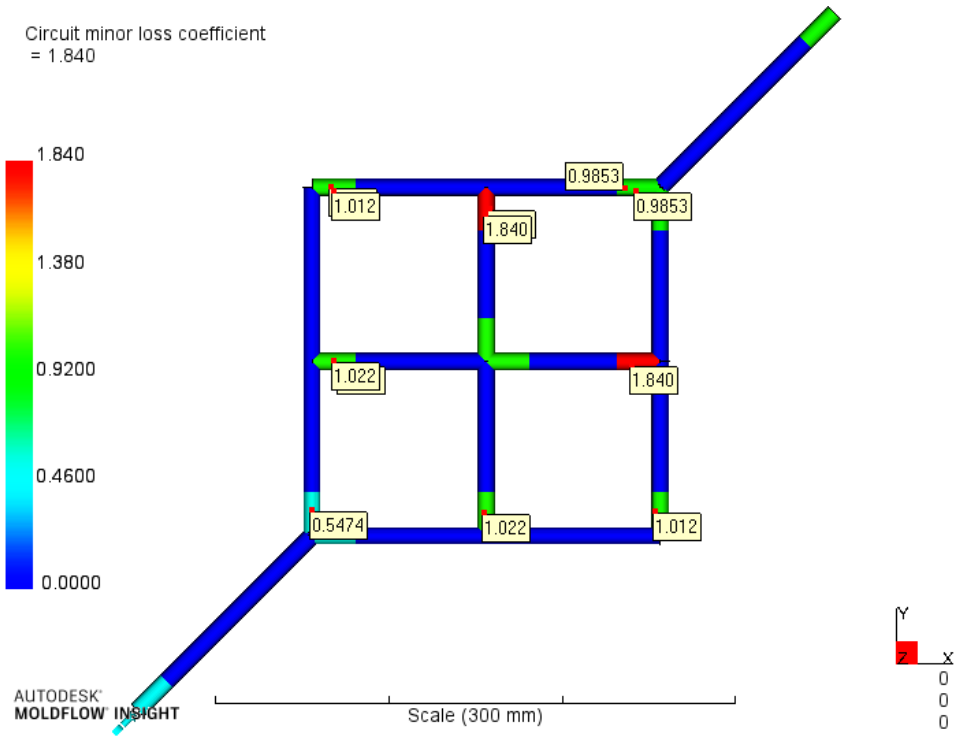


Figure 18 Minor loss calculation for bends and Tees

Conclusions

The Coolant flow analysis allows users to optimize the heat exchange in the mold through careful design of the cooling channels without the requirement of having to model the part together with the mold.

The users can now focus on the cooling system design without the overhead of the part and mold models in the analysis that take a long time to model and solve. When doing a Coolant flow analysis, thermal boundary conditions can be set on channel elements to evaluate the heat removing capability of the mold circuit design

Users now have more control over the selection of the minor loss factors and pipe friction loss formulations that can be used in a Cool or Cool (FEM) analysis. The option of taking gravity into account and the option of using a pump element with an exact pump curve to model the pump performance is also supported in midplane, dual domain and 3D Cool solutions.

The existing theory, implementation and demonstration of the new options have been discussed and demonstrated.

.

References

1. S. V. Patankar and D. B Spalding, A calculation procedure for heat mass and momentum transfer in three-dimensional parabolic flows', Int. J. Heart and Mass Transfer, 15, 1787-1806 (1970)
2. Greyvenstein, Gideon P. (1991), Proceedings of the second national symposium on computational fluid dynamics, Riviera Sun Hotel, Vereeniging, 24-27 June 1991.
3. Greyvenstein, GP, Laurie DP, A segregated CFD approach to pipe network analysis. International Journal for Numerical Methods in Engineeering 1994; 37:3685-3705.
4. Larock, Bruce E., Jeppson, Roland W., Watters, Gary Z.,(2000), Hydraulics of Pipeline Systems, CRC Press LLC
5. Shames, Irving H. (1992), Mechanics of fluids, third edition, McGraw-Hill, Inc
6. Idelchick, I.E. (2008), Handbook of Hydraulic Resistance, 3rd Edition, Jaico Publishing house.
7. Miller, Donald. S, Internal flow systems.
8. Cengel, Yunus A., Cimbala, John M., Turner, Robert H., (2008) Fundamentals of Thermo-Fluid Sciences, McGraw-Hill, Inc
9. Menon, E. Shashi. (2005), Piping calculations Manual, McGraw-Hill, Inc

Autodesk and Moldflow are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document.

© 2018 Autodesk, Inc. All rights reserved.