Autodesk® Moldflow® Insight 2012

AMI Cool Analysis Products

Revision 1, 22 March 2012.

Contents

Cool analysis products

1

The Cool analysis products are heat transfer simulations used to analyze the flow of heat in plastic injection molds. These products are available using the Thermoplastics Injection Molding process.

The Cool analysis products include:

Steady state cool - for optimizing the design of the part

The steady-state cool simulation analyzes the both the temperature of the part and the temperature of the mold, calculating a cycle averaged temperature distribution in the mold, in order to optimize various aspects of the design of the part, including cooling time, cycle time, part design, and mold design.

Steady state cool analyses are supported by the following mesh types:

- Midplane
- Dual Domain
- 3D

  There are two different 3D solutions from which to select; Cool, using the Boundary element method or Cool (FEM), using the finite element method.

Transient cool - for analyzing temperature distributions in the mold

Transient cool analyses are used to simulate the change in mold temperature with time.

Transient cool analyses are supported by the following mesh types:

- 3D - using the Cool (FEM) solution.

NOTE: When switching from one Cool analysis product to the other, that is from Cool to Cool (FEM), or back, in each case the mold mesh will need to be regenerated.

Features of Cool analysis products

2

The Cool analyses are heat transfer analysis products designed to analyze flow of heat in plastic injection molds. The principle goals of these analyses are to determine temperatures in the plastic filled cavity and throughout the mold, and to determine the cooling time.

The design engineer has two main concerns when cooling a mold:

- The mold is essentially a heat exchanger which must be capable of extracting heat at the required rate.
- The part must cool uniformly, in order to ensure a high quality product with minimum distortion.

The two factors, adequate heat extraction and uniform cooling of the mold, form the basis of the Cool analysis design philosophy.

In addition to these primary results, cooling network flow parameters, such as pressure or flow rate requirements, are also produced. Network analysis also provides information on pumping requirements for a given circuit and coolant combination.

A feature of the Cool analysis products is that they interface to the Fill+Pack analyses. This enables the Fill+Pack analysis to recognize the effects of local hot and cold spots arising from a given cooling situation. Warp, in turn, considers the cooling effects carried onto the Fill+Pack analysis in order to compute the impact of differential temperature distributions on part warpage .

Cool analysis results are based on the assumption that the model is initially filled with material at the melt temperature.

Temperatures can be used to understand the effectiveness of:

- Cooling channel location.
- Coolant inlet temperature.
- Coolant flow parameters.
- Mold inserts.
- Mold parting planes.
- Mold external surfaces.
- Part design on uniformity of part cooling.
- Mold temperature distributions.
- Cycle time.

Preparing the model for Cool analysis

3

This topic provides you with a comprehensive list of all of the modeling tasks for a Cool analysis.

Compulsory modeling tasks

- A meshed part and mold model.
- Mold material must be assigned as same value for mold, circuits and parting plane, but may be a different value for inserts.
- Circuit inlet points (coolant inlets) must be assigned to each cooling circuit.
- Coolant type must be assigned for each circuit.
- Coolant inlet temperature must be assigned for each circuit
- An HTE value must be assigned to bubblers or baffles (where they exist).
- The flow rate or the Reynolds number for each cooling circuit must be assigned to each circuit if you intend to analyze the cooling network using either the flow rate or minimum Reynolds number options.
- No elements should intersect with the mold boundary elements.
- Outer mold boundaries should be modeled. This results in a more accurate heat transfer model.

Optional modeling tasks

- Interface conductance may be assigned to a mold insert or a parting plane. If no conductance is assigned, the program will assume a high conductance across the interface.

Specified and automatic cooling time in a Cool analysis 4

When you run a Cool analysis in Autodesk Moldflow Insight, you can specify whether to set the injection + packing + cooling time manually (specified), or as automatic.

The following topic describes the differences between a Cool analysis with a manual or automatic time set.

NOTE: This option is only available if you are running an analysis sequence that begins with Cool.

Specified analysis

A specified analysis is used when you want to examine temperature distributions for a given cavity, mold and processing conditions. The result is the mold temperature distribution given by the fixed variables of melt temperature, coolant selection and coolant flow rates.

It is also the analysis required when a design is being taken through to Warp analysis as plastic processing conditions and cycle time are generally fixed by the molding requirements.

Cooling of the plastic occurs from the commencement of the filling phase. However, the “Injection + packing + cooling time” value specified is the time required to cool the part sufficiently for ejection. The material at the center section of the part wall reaches its transition temperature and becomes solid during this time.

Automatic analysis

An automatic analysis adjusts processing parameters to optimize the cooling time required to achieve a target average mold temperature specified in the **Process Settings Wizard** and the specified percentage freeze level in the plastic part.

The automatic analysis calculates the cycle time based on two parameters:

- Achieving a target mold temperature within 1°C.
- The percentage of the part that is frozen.

Because of these assumptions, the automatic analysis should not be run unless the cooling system has been optimized, and the average cavity temperature is less than the target mold temperature.

The target mold temperature is the **Mold surface temperature** entered in the **Process Settings** wizard.

The percentage plastic frozen is the ratio of the volume of plastic that is frozen in the part to the volume of the part. During an automatic analysis, a plastic state of freeze target is required. The default value is 100 percent frozen. If this yields cooling times that are too long, for example, where a thick section is present, reduce the **Frozen percentage at ejection** value. This will reduce the calculated cooling time, whilst maintaining adequate freezing in the critical areas of the cavity.

Cool analysis for part optimization

5

Steady state cool analyses provide an accurate way to analyze the cycle-averaged temperature distribution for a given cycle. The results are then used to optimize the design of your mold and ensure the production of a quality part.

The steady-state cool simulation analyzes the both the temperature of the part and the temperature of the mold, calculating a cycle-averaged temperature distribution in the mold, in order to optimize various aspects of the design of the part, including cooling time, cycle time, part design, and mold design.

Midplane When a Midplane Cool analysis is performed, the solver calculates heat loss in the x and y directions, and makes no estimate in the z-direction. The calculation uses a semi-infinite slab calculation in the part to calculate the fluxes and temperature distribution. These fluxes are then used as boundary conditions for the boundary element solution that calculates the surface temperatures of the mold.

Dual Domain When a Dual Domain Cool analysis is performed, the solver calculates heat loss in the x and y directions, and makes an estimate of the heat loss in the z-direction. The calculation uses a semi-infinite slab calculation in the part to calculate the fluxes and temperature distribution. These fluxes are then used as boundary conditions for the boundary element solution that calculates the surface temperatures of the mold.

3D When a 3D Cool analysis is performed, the solver obtains a full three-dimensional transient finite-element solution for the temperatures of the part, which is used for the calculation of the heat flux into the mold. There are two different solutions, Cool and Cool (FEM), available for calculating the temperature in the mold.

NOTE: When switching between the Cool and Cool (FEM) solvers, the mold mesh will be deleted and must be regenerated each time.

- *Cool* - The heat fluxes from the part are used as boundary conditions for the steady-state *boundary element* solution that calculates the surface temperatures of the mold. The boundary element method (BEM) determines the temperature on all surfaces of the mold, that is the outer surface, the part and the cooling channel surfaces, then uses the boundary element integrals to calculate the internal temperatures of the mold. This provides an accurate representation of the temperature and enables you to optimize the placement, quantity and operating conditions of cooling channels in the mold. This option provides a similar solution in the mold as the Cool (FEM) option, when the conduction solver is selected in the part.

NOTE: To run a Cool analysis it is not necessary to model a mold; only the cooling circuits are required.

- *Cool (FEM)* - The heat fluxes from the part are used as boundary conditions for the steady-state *finite-element* solution that calculates the temperature through the depth of the mold. This provides the temperature at every node through the mold, and enables you to optimize the placement, quantity and operating conditions of cooling channels in the mold.

NOTE: To run a Cool (FEM) analysis, it is necessary to model a mold to surround part and cooling circuits.

There are two solvers to choose from for calculating the temperature distribution in the part:

- Conduction solver - this is a fast solver, that only considers heat conduction. This solver provides a similar result to Cool (BEM), in a shorter timeframe.
- Flow solver - this solver solves the entire flow solution in the part, passes the data across for calculation of the mold temperature distribution, then takes the temperature information from the mold and recalculates the entire flow solution in the part. This process is reiterated many times over, until the results converge. This solver is slower than either the conduction solver, or Cool (BEM), but the results can be used to capture shear heating effects from flow, that are caused by heat fluxes from the part into the mold.

Cool analysis

The Cool analysis is used to analyze the flow of heat in the mold.

Setting up a Cool analysis

The following table summarizes the setup tasks required to prepare a Cool analysis.

The setup tasks below are for thermoplastic materials.

Setup task	Analysis technology
Molding processes	
Creating cooling channels manually	

Setup task	Analysis technology
Meshing the model	
Mesh orientation	
Checking the mesh before analysis	
Mold material	
Analysis sequence	
Selecting a material	
Injection locations	
Process settings	
Coolant inlets	
Cooling circuit parameters	

Optional setup tasks

Setup task	Analysis technology
Modeling bubblers	
Modeling baffles	
Mold Surface Wizard	
Create Inserts tool	
Part Insert dialog	
Editing heat transfer effectiveness properties	
Editing interface conductance properties	

Running a Cool analysis

Run a Cool analysis to analyze the flow of heat in plastic injected molds.

After the analysis has finished, you can examine the Confidence of Fill and Quality Prediction results, as well as Fill Time, Injection Pressure, Pressure Drop and Flow Front Temperature results.

1 Open a model.
2 Set the parting plane.
3 Define the mold dimensions and select the mold material.
4 Set coolant inlets.

5 Click **Home tab > Molding Process Setup panel > Analysis Sequence**.

6 In the **Select Analysis Sequence** dialog, select **Cool** and then click **OK**.

7 Double-click **Start Analysis!** from the **Study Tasks** pane, or **Home tab > Analysis panel > Start Analysis**.

NOTE: Click **Home tab > Analysis panel > Job Manager > Abort Job** to abort the analysis.

Setting up an *Averaged within cycle* Cool analysis (Cool (FEM))

The following table summarizes the setup tasks required to prepare an **Averaged within cycle** cool analysis, using the Cool (FEM) solution.

Setup Task	Analysis Technology
Molding processes	
Analysis sequence	
Meshing the model	
mesh orientation	
Checking the mesh before analysis	
injection locations	
Selecting a material	
Importing a mold on page 18 or *Mold Surface Wizard* on page 30	
Mold material	
cooling circuits	
coolant inlets	
cooling circuit parameters	
process settings	

Optional setup tasks

Setup Task	Analysis Technology
Modeling bubblers	
Modeling baffles	
Editing heat transfer effectiveness properties	

Setup Task	Analysis Technology
Editing interface conductance properties	

Running an *Averaged within cycle* Cool analysis (Cool (FEM))

Detailed instructions for each of these steps are available under relevant sections. Use the help search feature to locate pages of interest.

1. Import or model a part and associated feed system.
2. Click **(Home tab > Molding Process Setup panel > Analysis Sequence)** and select a Cool (FEM) analysis sequence.
3. Click **(Mesh tab > Mesh panel > Generate Mesh)** and mesh the part and feed system.
4. With only the meshed part layer visible, assign appropriate properties to the part and the various feed system components.
5. Set the injection location on the meshed part.
6. Import or model the mold and associated cooling system.
7. Assign appropriate properties to the mold and the various cooling circuit components.
8. Set the coolant inlets on the channels layer.
9. Make sure the meshed part layer, the cooling channels layer and the mold layer are all visible, then click **(Mesh tab > Mesh panel > 3D Mold Mesh)** and mesh the mold and cooling system.
10. Click **(Home tab > Molding Process Setup panel > Process Settings)** and set the **Mold temperature options** to **Averaged within cycle**.
11. Click **Cool (FEM) Solver parameters** and select which solver you would like to use from **Steady, part heat flux calculation**.
12. Click **OK** twice to exit the **Process Setting Wizard**.
13. Click **(Home tab > Analysis panel > Start Analysis)** to start the analysis.

NOTE: Click **(Home tab > Analysis panel > Job Manager > Abort Job)** to abort the analysis.

Cool analysis

Use this dialog to specify settings for a Cool analysis.

Process Settings Wizard dialog—Cool settings

This page of the **Process Settings Wizard**, which can be accessed by clicking (**Home tab > Molding Process Setup panel > Process Settings**), is used to specify the Cool analysis related process settings for the selected analysis sequence.

NOTE: Some of the items listed below may not be available on the current dialog. This is dependent on the analysis technology, molding process and analysis sequence selected.

Melt temperature	The temperature of the molten plastic, or melt, as it starts to flow into the cavity.
Mold-open time	Specify the time taken from the completion of one molding cycle, to the beginning of the next.
Injection + packing + cooling time	Injection + packing + cooling time is equivalent to the total cycle time minus the mold-open time. You can either specify this time directly, or calculate this time automatically during the analysis.
Edit target ejection criteria	Used to specify the part ejection criteria for a Cool analysis. Edit the variables as appropriate.
Cool Solver Parameters	Displays the Cool analysis related solver parameters for the selected analysis sequence.
Advanced options...	Displays the advanced options for the analysis.

Cool Solver Parameters dialog

This dialog is used to specify the Cool analysis related solver parameters for the selected analysis sequence.

To access this dialog, ensure that you have selected an analysis sequence that includes Cool, click (**Home tab > Molding Process Setup panel > Process Settings**), if necessary click **Next** one or more times to navigate to the **Cool Settings** page of the Wizard, then click **Cool solver parameters**.

Mold temperature convergence tolerance	Convergence tolerances apply to the % change in a function value from one iteration to the next, and are used to identify when a solution has converged.

Maximum number of mold temperature iterations	Select the maximum number of mold-temperature iterations for the analysis to attempt.
Number of time steps for 3D cooling flux calculation	A larger number of time steps will lead to more accurate results, but will require a longer solution time.
Include runners in automatic cooling time calculations	Specify if the runner system is to be included when automatic cooling time is calculated.
Use aggregated mesh solver	Mesh aggregation is a feature that reduces the number of elements of the model internal to the solver, allowing for much faster Cool analysis times.
Calculate internal mold temperatures	When selected, this option enables the creation of data for the **Temperature, internal mold** result.

Cool (FEM) Solver Parameters

This dialog is used to specify whether to use the **Conduction solver** or whether to run the **Flow analysis on every iteration** solver. The Conduction solver will be faster, but the Flow solver may provide a more accurate simulation.

To access this dialog, ensure that you have run the 3D mesher on your part, you have selected an analysis sequence that includes Cool (FEM), and in the **Process Settings** you have selected **Averaged within cycle** from the **Mold temperature options** drop-down menu. Then click on the **Cool (FEM) Solver Parameters** button.

Once you have selected the solver to run, you can specify the following solver parameters:

Steady, part heat flux calculation	Select the solver you would like to use to calculate the heat flux. The **Conduction solver** is faster, but the **Flow solver** provides more accurate results.	Default value: Conduction
Number of time steps	This applies to the temperature in the part. Specify how often you would like the conduction solver to determine the temperature of the part, during the **Injection + packing +cooling time** period you specified in the Process Settings Wizard.	For Conduction solver only
Process controller	Select and edit a process controller to control the injection molding process during the analysis.	For Flow solver only

Mold temperature convergence tolerance	Enter a value between the limits specified. The convergence tolerance applies to the % change in mold temperature, from one iteration to the next. The lower the tolerance the more accurate the result, but the longer the analysis.	Default value: 0.10000
Maximum number of mold temperature iterations	Select the maximum number of mold-temperature iterations you would like the analysis to perform. If the result does not converge before this maximum value, you will receive an error message.	Default value: 50
Number of threads for parallelization	Select one of the drop-down options to configure parallel computation. This will improve solution speed, especially for large models.	Default value: Automatic

Part Ejection Criteria dialog

This dialog is used to specify the part ejection criteria for a Cool analysis with automatic calculation of the required cooling time. These inputs are used by the Cool and Cool (FEM) solvers to determine the optimum cooling time for the part.

To access this dialog, ensure that you have selected an analysis sequence that includes Cool or Cool (FEM), click (**Home tab > Molding Process Setup panel > Process Settings**), and set the **Injection + packing + cooling time** option to **Automatic**, then click **Edit target ejection criteria**.

When running an automatic cycle time, the solver initially tries to determine the time it takes to achieve the average **Mold surface temperature**. If this cannot be achieved, then the cycle time will be determined by using the time taken for the **Minimum part percentage at ejection temperature** to fall below the **Ejection temperature**.

For a Midplane or Dual Domain analysis, the cooling time is determined by the time taken for *each element* to reach at least the specified minimum part percentage frozen at ejection time.

A 3D analysis bases the cooling time on the *overall* part volume percentage frozen.

Mold surface temperature	The temperature of the mold at the plastic-metal interface, where the plastic touches the mold. The mold temperature cannot be higher than the ejection temperature.

Ejection temperature	The temperature at which a material is rigid enough to withstand ejection, without permanent deformation or severe marking from the mold ejector pins.
Minimum part percentage at ejection temperature	In order for the Cool analysis to automatically calculate the cooling time, the program needs to know the minimum part volume that must be frozen, before the part can be ejected.

Boundary element method derivation

Cool is a true 3D mold cooling analysis product. It uses a numerical method developed from BEM (Boundary Element Method). From a physical point of view, BEM treats all boundaries as heat sources (gain / loss heat) during the solution.

The temperature in the mold is determined by combining the influence from all sources.

The equilibrium temperature field of a 3D mold can be represented by Laplace's equation:

$$\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} = \nabla^2 T = 0 \quad p \in \Omega$$

where Ω includes the inside and surface area of a mold. The above equation refers to a specific point in that area, p, with boundary conditions unified as:

$$q = -k \frac{\partial T}{\partial n} = \bar{h}(T - T_\infty) \quad p \in S$$

where *T* represents the temperature, $\frac{\partial}{\partial n}$ denotes the outward normal derivative on the mold boundary, *k* is the thermal conductivity of the mold material, $\bar{h}$ is the equivalent heat transfer coefficient on the mold boundary and T_∞ the equivalent temperature of the ambient environment.

To understand how BEM applies all boundary conditions to the solution of the mold temperature field, let us start with the weighted residual expression:

$$\int_\Omega T^* \nabla^2 T \, d\Omega = 0$$

Where T* is the weighting function.

By making use of Green's second identity, *Equation 3* on page 14 can be transformed into the following form:

$$\int_\Omega \left(T^* \nabla^2 T - T \nabla^2 T^*\right) d\Omega = \int_S \left(T^* \frac{\partial T}{\partial n} - T \frac{\partial T^*}{\partial n}\right) dS$$

Choosing T^* as the fundamental solution of *Equation 1* on page 14 defined by:

$$\nabla^2 T^*(P, Q) = \delta(P, Q)$$

where $\delta(P, Q)$ is a Dirac delta function. For a 3D mold, this can be described as:

T P , Q = 1 4 π r P , Q

where *P* and *Q* are two points in the space and *r(P,Q)* represents the distance between the two points, then *Equation 4* on page 14 can be simplified as:

14π∫1r,QTQ∂n−TQ1/r,Q∂ndS=C

where

C=∫ΩΩ=∫Ω,QdΩ=1P0PCsP

Cs is a constant in proportion to the interior solid angle.

Now *Equation 7* on page 15 only has boundary integrations. So if we divide all mold surfaces (*S*) into *n* elements and assume the temperature and temperature gradient is constant over each boundary element, then *Equation 7* on page 15 can be discretised into the following form:

C+14πej=1nTejjk1/r∂ndS+qj1rdS=0

Herejk1/r∂ndS=Hei is the temperature influence term (or so-called *H* term) and represents the influence strength of temperature on element ei to point *P*.

j1r,QdS=Gei is the heat flux influence term (or so-called *G* term) and represents the influence strength of heat flux input on element ei to point P.

Suppose Pi is the centroid of element *i*. If we substitute *P* in *Equation 9* on page 15 with Pi, then we can get *n* linear equations as:

∑ei=1nTeiHeii+∑ei=1nqeiGeii=0

Cool analysis for transient mold temperature investigations

6

During an injection molding cycle, the mold temperatures, especially those in contact with the part, will heat up and cool down slightly around the average temperature. Typically, a mold will be hotter during the filling stage, and cooler during the pressurization and cooling stages. The Transient cool analyses enable you to see how the mold temperature varies with time.

When a 3D transient cool analysis is performed, the Cool (FEM) solver obtains a full three-dimensional transient finite-element solution for the temperatures of the part. This solution is used for the calculation of the heat flux into the mold.

Cool (FEM) The heat fluxes from the part are used as boundary conditions for the transient *finite-element* solution that calculates the temperature through the depth of the mold. This provides the temperature at every node through the mold, and enables you to optimize the placement, quantity and operating conditions of cooling channels in the mold.

NOTE: To run a Cool (FEM) analysis, it is necessary to model a mold. This can be accomplished in a 3rd party CAD program and imported, or it can be modeled using the **Mold Surface Wizard.**

There are two solvers to choose from for calculating the temperature distribution in the part:

- *Conduction solver* - this is a fast solver, that only considers heat conduction. This solver provides a similar result to Cool (BEM), in a shorter timeframe.
- *Flow solver* - this solver solves the entire flow solution in the part, passes the data across for calculation of the mold temperature distribution, the takes the temperature information from the mold and recalculates the entire flow solution in the part. This process is reiterated many times over, until the results converge. This solver is slower than either the conduction solver, or Cool (BEM), but the results can be used to capture shear heating effects from flow, that are caused by heat fluxes from the part into the mold.

NOTE: When switching between the Cool and the Cool (FEM) solvers, the mold mesh will be deleted and you will have to regenerate it each time.

Cool analysis for transient mold temperature investigations

The transient cool analyses use the finite element method for calculating results. To run a transient cool analysis, you will need to select an analysis sequence that includes a Cool (FEM) analysis.

Setting up a Transient Cool analysis

The following table summarizes the setup tasks required to prepare a Transient Cool analysis. Transient cool analyses can only be performed if you have used a 3D mesh.

The setup tasks below are for thermoplastic materials.

Setup Task	Analysis Technology
Molding processes	
Selecting an *Analysis sequence*	
Meshing the model	
Repairing *mesh orientation*	
Checking the mesh before analysis	
Setting *injection locations*	
Selecting a material	
Importing a mold on page 18 or *Mold Surface Wizard* on page 30	
Selecting a *Mold material*	
Creating *cooling circuits*	
Setting *coolant inlets*	
Editing *cooling circuit parameters*	
Editing *process settings*	

Optional setup tasks

Setup Task	Analysis Technology
Modeling bubblers	
Modeling baffles	
Editing heat transfer effectiveness properties	

Setup Task	Analysis Technology
Editing interface conductance properties	

Importing a mold

Different file formats can be imported into Autodesk Moldflow Insight for analysis. This section provides general guidelines for working with the supported file formats so that data is translated appropriately.

The following molds can be imported:

- CAD/STL molds
- molds from Autodesk Inventor Tooling (AIT)

  Although molds created in Autodesk Inventor Professional (.sat) can be imported into Autodesk Moldflow Insight, you will have to create curves to represent the feed system and cooling channels, separately. For molds created in AIT (.sat), both the mold body and the curves will be imported during the import, removing the subsequent requirement to create curves for the feed system and cooling channels.

If you do not have a mold to import, you can use the *Mold Surface Wizard* on page 29 to create a mold around your part, feed system and cooling system.

Importing a mold

Once you have imported all the necessary components, you will need to assign them the correct properties.

Importing a CAD/STL mold model

You can run a finite element cool analysis on an existing a CAD/STL mold model. If you want to import a CAD/STL mold, make sure that the feed system and cooling channels have already been modelled, according to your CAD/STL design.

To import a CAD model, you will need Autodesk Moldflow Design Link 2012, or later, installed on your computer.

The first steps 1-4 will instruct you on how to import your model and feed system. Steps 5-7 apply to importing the mold. The same procedure applies to all the components you need to import.

1 Click **(Home tab > Import panel > Import)**.
2 Browse to the folder containing your modeled part and runner system, select it and click **Open**.
3 In the **Import** dialog, select **Solid 3D** from the mesh type drop-down menu.

4 When importing a CAD model, confirm that the radio button associated with **Direct Import using Autodesk Moldflow Design Link** is selected, then press **OK**.
The unmeshed model will appear in your Project.

5 Click **(Home tab > Import panel > Add)** to add the mold.

6 Browse to the folder containing your mold, select the mold and click **Open**.
The type of mesh will have been decided, based on your initial model.

7 Confirm that the radio button associated with **Direct Import using Autodesk Moldflow Design Link** is selected, then press **OK**.
The unmeshed mold will appear in your Project.

8 Repeat this process for all mold components, such as channels.
You will now need to *Generating the mold mesh* on page 33

Importing a model from Autodesk Inventor Tooling, including mold, part, feed system and cooling channels

Before you can import your model into Autodesk Moldflow Insight, you must have modeled one in Autodesk Inventor Tooling and exported it to an accessible location.

NOTE: Make sure you have Autodesk Moldflow Design Link 2012 installed on your computer, before you import your model into Autodesk Moldflow Insight.

Steps 1-5 below refer to exporting a model from Autodesk Inventor Tooling. Steps 6-9 explain how to import it into Autodesk Moldflow Insight.

1 Create the model in Autodesk Inventor Tooling, according to the instructions in that program.

2 In Autodesk Inventor Tooling, click **(Mold Layout tab > Tools panel > Export)** and select **Autodesk Moldflow Insight** from the **Export Models** section of the **Export** dialog.

3 Navigate to your preferred destination folder and change the name of your mold model, if desired.

4 Click **Export**.
All the mold details will be removed, leaving only the mold, part, feed system and cooling channels.

5 In the **Export Options** dialog, all components will be pre-selected for export. De-select any components you do not want to export from both the Mold block tab and the Cooling system tab, then click **OK** to start the export process, and **OK** to close the export successful dialog.
The original model will be in AIT. The model for export will not be saved in AIT.

6 In Autodesk Moldflow Insight, click **(Home tab > Import panel > Import)**.

7 Browse to the folder containing your AIT model, select it and click **Open**.

8 In the **Import** dialog, select **Solid 3D** from the mesh type drop-down menu.

9 Confirm that the radio button associated with **Direct Import using Autodesk Moldflow Design Link** is selected, then press **OK**.
The unmeshed model will appear in your Project. You will now need to *Generating the mold mesh* on page 33

Importing a mold from Autodesk Inventor Tooling to add to an existing part.

Before you can import your model into Autodesk Moldflow Insight, you must have modeled one in Autodesk Inventor Tooling and exported it to an accessible location.

NOTE: Make sure you have Autodesk Moldflow Design Link 2012 installed on your computer, before you import your model into Autodesk Moldflow Insight.

Steps 1-5 below refer to exporting a mold from Autodesk Inventor Tooling. Steps 6-8 explain how to add it to your study.

1 Create the mold in Autodesk Inventor Tooling, according to the instructions in that program. Include a feed system and cooling channels if necessary.

2 In Autodesk Inventor Tooling, click **(Mold Layout tab > Tools panel > Export)** and select **Autodesk Moldflow Insight** from the **Export Models** section of the **Export** dialog.

3 Navigate to your preferred destination folder and change the name of your mold model, if desired.

4 Click **Export**.
All the mold details will be removed, leaving only the mold, and any feed system and cooling channels.

5 In the **Export Options** dialog, all components will be pre-selected for export. De-select any components you do not want to export from both the Mold block tab and the Cooling system tab, then click **OK** to start the export process, then **OK** again to close the export successful dialog. The original model will be in AIT. The model for export will not be saved in AIT.

6 With your relevant study open in Autodesk Moldflow Insight, click **(Home tab > Import panel > Add)** to add the mold to your part study.

7 Browse to the folder containing your mold, select the mold and click **Open**.
The type of mesh will have been decided, based on your initial model.

8 Confirm that the radio button associated with **Direct Import using Autodesk Moldflow Design Link** is selected, then press **OK**.

The unmeshed mold will appear in your Project. You will now need to *Generating the mold mesh* on page 33

Assigning properties to mold components

Once the mold and all its components have been imported, they must be assigned the correct properties. The best way to do this is to select one layer at a time and deselect all others.

1. Select the mold layer and make sure all other layers are deselected.
2. Click ((**View tab > Navigate panel > Select**)), and then click on the mold to select it.
3. Click **(Geometry tab > Properties panel > Assign)** and select **Mold block (3D)** from the **New** drop-down list.
   The Mold block (3D) dialog opens.
4. Edit the entries as appropriate, then click **OK** twice to close all dialogs.
5. Repeat this process with all the other components, including the part, the feed system and the cooling channels.

Importing a mold

Once you have imported a mold, it is necessary to assign to it the appropriate property.

Mold block (3D) dialog

This dialog is used to edit the properties of mold block elements for transient cool analyses.

To access this dialog, click **(Geometry tab > Properties panel > Assign)** and select **Mold Block (3D)** from the **New** drop-down list.

Dialog Properties	Comment
Mold material	Select **Use global settings in advanced options**, if you would like to use the mold material values defined by the **Advanced options** of the **Process Settings Wizard**. To choose a different material, select **Select a different mold material**, click **Select**, twice, and select one from the table.
External heat transfer coefficients	Select **Air** for mold block surfaces surrounded by air. Assign specific values to surfaces in contact with metal, for example where the mold attaches to the platen.
Mold block conductance	A measure of conductance between two plates. The default is 30,000 W/m^2*C, representing perfect clamping. Decrease

Dialog Properties	Comment
	this value to reflect less than perfect clamping.
Initial temperature for production start-up	This property is only used for Production Start-up analyses. Enter the temperature of the mold before the analysis starts.
Name	The collection of property values defined in this dialog are saved to a property set with the description shown in the Name box. You can leave the default or type in a different name.

Mold insert (3D) dialog

This dialog is used to edit the properties of mold insert (3D) elements for Cool analyses, when the Cool (FEM) solver has been selected.

To access this dialog, click **(Geometry tab > Properties panel > Assign)** and select **Mold insert (3D)** from the **New** drop-down list. You can also access this dialog from **(Geometry tab > Properties panel > Edit)** if you would like to edit the assigned properties.

Dialog Properties	Comment
Mold material	Select **Use global settings in advanced options**, if you would like to use the mold material values defined by the **Advanced options** of the **Process Settings Wizard**. To choose a different material, select **Select a different mold material**, click **Select**, twice, and select one from the table.
External heat transfer coefficients	Select **Air** for mold inserts in contact with air. Assign specific values to surfaces in contact with metal, for example where the mold attaches to the platen.
Mold block conductance	A measure of conductance between two plates. The default is 30,000 W/m^2*C, representing perfect clamping. Decrease this value to reflect less than perfect clamping.
Initial temperature for production start-up	This property is only used for Production Start-up analyses. Enter the temperature of the mold before the analysis starts.
Name	The collection of property values defined in this dialog are saved to a property set with the description shown in the Name box. You can leave the default or type in a different name.

Selecting a Transient cool analysis

There are two transient cool analysis products for studying mold temperature fluctuations; transient within cycle and transient from production start-up. In both cases, the Cool (FEM) solver determines the results using the finite element method for calculation.

Transient within cycle cool analyses

This solver simulates mold temperature deviations from the cycle average. The typical injection molding cycle consists of the filling, packing, cooling and mold open stages.

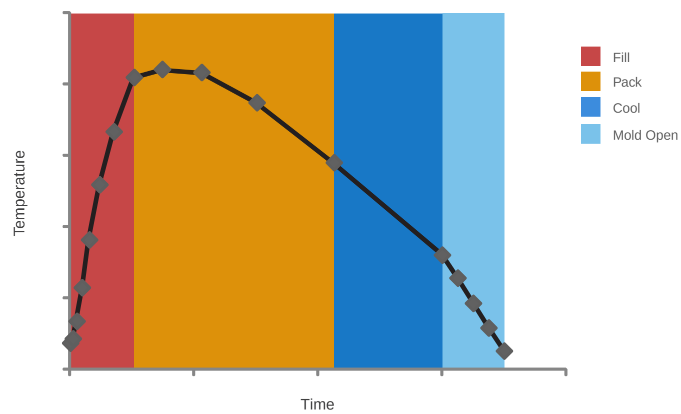


Figure 1: Change in temperature of mold during an injection molding cycle

During the filling stage the high temperature molten plastic is injected into the mold cavity at high pressure. This causes the temperature of the mold to rise. Following injection, the molten plastic cools down as a result of the cooling channels in the mold, until it reaches a cool enough temperature that the plastic part can be ejected from the mold at the commencement of the mold open stage. As a result, the temperature in the mold also decreases. As shown in the graph above, therefore, the mold temperatures rise during the filling and packing stages but drop during the cooling and mold open stages. The transient cool solver can be used to simulate the mold temperatures at different stages of the injection molding cycle.

Transient from

This solver simulates the mold temperature variations from the initial production start-up cycle when the mold is cold, until the mold temperature stabilizes and reaches its optimum operating conditions.

production start-up cool analyses

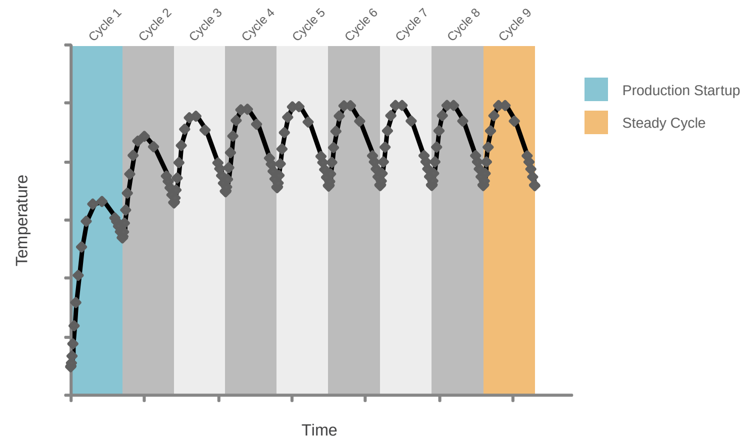


Figure 2: Change in temperature of mold from one cycle to the next during a production start-up routine

When an injection molding machine is commissioned to manufacture a product, it may require many cycles before the mold reaches its optimum operating conditions, or stable cycle. Once the stable cycle has been reached, the mold temperature within a cycle still changes, as shown in the transient within cycle graph, but is constant between cycles as shown in the graph above. Until the stable cycle has been attained, the quality of the part cannot be guaranteed. The transient from production start up solver simulates the process leading up to the stable cycle and helps predict how many cycles will be required.

The Cool (FEM) solver can also be used for a *Cool analysis for part optimization* on page 6. This provides a similar solution to the Cool (BEM) solver if the conduction solver is selected, but in a shorter timeframe. If the flow solver is selected, the results can be used to capture shear heating effects from flow, that are caused by heat fluxes from the part into the mold.

Selecting a Transient cool analysis

Before you can select a transient cool analysis, you must have selected an analysis sequence that contains a Cool (FEM) analysis.

Selecting a Transient within cycle analysis

Make sure that you have a 3D model and have selected Thermoplastics Injection Molding.

1 Click (**Home tab > Molding Process Setup panel > Analysis Sequence**), and select the Cool (FEM) analysis of interest.

NOTE: Transient cool analyses are only available for a Thermoplastic Injection molding process, using the finite element method (FEM) solver.

---

2 Notice that a **Create Mold Mesh** requirement appears in the **Study Tasks** pane, under the **Cooling Circuit(s)**.

3 Click (**Home tab > Molding Process Setup panel > Process Settings**) to open the **Process Setting Wizard - Cool (FEM) Settings** dialog.

4 Select **Transient within cycle** from the **Mold temperature options** drop-down list.

5 Click on the **Cool (FEM) Solver Parameters** button and select whether to use the **Conduction solver** or the **Flow solver** (Flow analysis on every iteration) from the **Transient, part heat flux calculation** drop-down menu.
The **Conduction solver** returns a faster result but the **Flow solver** may return a more accurate one.

6 Check the default values and edit them if necessary, then click **OK**.

NOTE: Generally, the default settings should suffice.

7 Click **Next**, if necessary, to proceed through the Process Settings Wizard dialogs, then click **OK** to exit.

Selecting a Transient from production start-up analysis

1 Click (**Home tab > Molding Process Setup panel > Analysis Sequence**), and select the Cool (FEM) analysis of interest.

NOTE: Transient cool analyses are only available for a Thermoplastic Injection molding process, using the finite element method (FEM) solver.

2 Notice that a **Create Mold Mesh** requirement appears in the **Study Tasks** pane, under the **Cooling Circuit(s)**.

3 Click (**Home tab > Molding Process Setup panel > Process Settings**) to open the **Process Setting Wizard - Cool (FEM) Settings** dialog.

4 Select **Transient from production start-up** from the **Mold temperature options** drop-down list.

5 Click on the **Cool (FEM) Solver Parameters** button and select whether to use the **Conduction solver** or the **Flow solver** (Flow analysis on every iteration) from the **Transient, part heat flux calculation** drop-down menu.
The **Conduction solver** returns a faster result but the **Flow solver** may return a more accurate one.

6 Check the default values and edit them if necessary, then click **OK**.

NOTE: Generally, the default settings should suffice.

7 Click **Next**, if necessary, to proceed through the Process Settings Wizard dialogs, then click **OK** to exit.

Selecting a solver for transient cool analyses

In order to select a solver for transient cool analyses, you must first select an analysis sequence that contains a Cool (FEM) analysis.

1 Click (**Home tab > Molding Process Setup panel > Analysis Sequence**), and select the Cool (FEM) analysis of interest.

NOTE: Transient cool analyses are only available for a Thermoplastic Injection molding process, using the finite element method (FEM) solver.

---

2 Click (**Home tab > Molding Process Setup panel > Process Settings**) to open the **Process Setting Wizard - Cool (FEM) Settings** dialog.

3 Click on the **Cool (FEM) Solver Parameters** button and select whether to use the **Conduction solver** or the **Flow solver** (Flow analysis on every iteration) from the **Transient, part heat flux calculation** drop-down menu.
The **Conduction solver** returns a faster result but the **Flow solver** may return a more accurate one.

4 Check the default values and edit them if necessary, then click **OK**.

NOTE: Generally, the default settings should suffice.

---

5 Click **Next**, if necessary, to proceed through the Process Settings Wizard dialogs, then click **OK** to exit.

Selecting a Transient cool analysis

Use this dialog to specify settings for a Transient Cool analysis.

Process Settings Wizard dialog—Cool (FEM) settings

This page of the **Process Settings Wizard**, which can be accessed by clicking (**Home tab > Molding Process Setup panel > Process Settings**), is used to specify the Cool analysis related process settings for the selected analysis sequence.

NOTE: Some of the items listed below may not be available on the current dialog. This is dependent on the analysis technology, molding process and analysis sequence selected.

Melt temperature	The temperature of the molten plastic, or melt, as it starts to flow into the cavity.
Mold-open time	Specify the time taken from the completion of one molding cycle, to the beginning of the next.
Injection + packing + cooling time	Injection + packing + cooling time is equivalent to the total cycle time minus the mold-open time. You can either specify this time directly, or calculate this time automatically during the analysis.
Edit target ejection criteria	Used to specify the part ejection criteria for a Cool analysis. Edit the variables as appropriate.
Mold temperature options	Select the type of simulation you would like to perform on the mold; Averaged within cycle, Transient within cycle, or Transient from production start-up
Cool (FEM) Solver Parameters	Displays the Cool analysis related solver parameters for the selected analysis sequence.
Advanced options...	Displays the advanced options for the analysis.

Cool (FEM) Solver Parameters

This dialog is used to specify whether to use the **Conduction solver** or whether to run a **Flow analysis on every iteration**. The Conduction solver will be faster, but the Flow solver may provide a more accurate simulation.

To access this dialog, ensure that you have run the 3D mesher on your part, you have selected an analysis sequence that includes Cool (FEM), and in the **Process Settings** you have selected one of the **Transient** analysis options from the **Mold temperature options** drop-down menu. Then click on the **Cool (FEM) Solver Parameters** button. The variables in the dialog will change depending upon the **Mold temperature options** you select in the **Process Settings Wizard**.

Once you have selected the solver to run, you can specify the following solver parameters:

Transient, part heat flux calculation	Select the solver you would like to use to calculate the heat flux. The conduction solver is faster, but the flow solver provides more accurate results.	Default value: Conduction

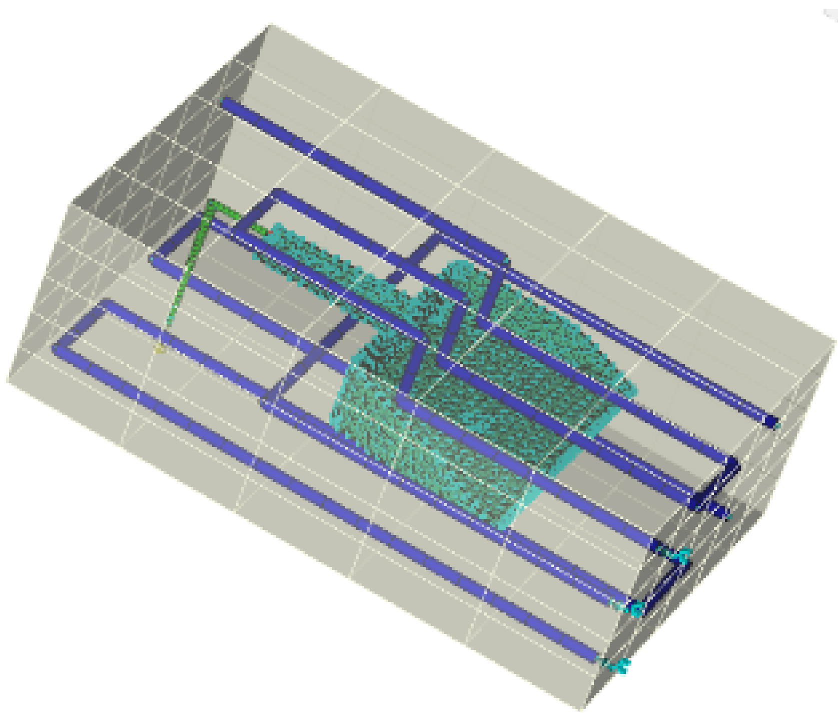
Number of time steps	This applies to the temperature in the part. Specify how often you would like the conduction solver to determine the temperature of the part, during the **Injection + packing +cooling time** period you specified in the Process Settings Wizard.	For Conduction solver only
Transient mold temperature convergence tolerance for each time step	Enter a value between the limits specified. The convergence tolerance applies to the change in mold temperature, from one iteration to the next, within a single time step. The lower the tolerance the more accurate the result, but the longer the analysis.	Default value: 0.01000
Maximum number of transient mold temperature iterations for each time step	Select the maximum number of mold-temperature iterations you would like the analysis to perform for each time step. If the result does not converge before this maximum value, you will receive an error message.	Default value: 50
Transient mold temperature convergence tolerance	Enter a value between the limits specified. The convergence tolerance applies to the change in mold temperature, from one iteration to the next. The lower the tolerance the more accurate the result, but the longer the analysis.	Default value: 0.1000
Maximum number of transient mold temperature cycles	Select the maximum number of mold-temperature iterations you would like the analysis to perform. If the result does not converge before this maximum value, you will receive an error message.	Default value: 100
Number of threads for parallelization	Select one of the drop-down options to configure parallel	Default value: Automatic

	computation. This will improve solution speed, especially for large models.

Mold Surface Wizard

To run a Transient Cool analysis, you will need to have modelled a mold with associated cooling channels, hoses, baffles etc. and any additional components, such as mold inserts. If you don't have a mold that you plan to import, you can use the **Mold Surface Wizard** to create a mold around the part, feed system and cooling channels.

Mold models defined through the **Mold Surface Wizard** must contain both cooling channels and a feed system. The feed systems can be represented by tetras or beam elements, but both the feed system and the cooling channels must be represented by underlying curves, in order to be included in the mold mesh generation.



NOTE: Existing feed systems and cooling channels that are represented by beams, without underlying curves, will not be considered.

NOTE: Tetrahedral elements must be fully enclosed by the mold surface. Beam elements may protrude.

Mold Surface Wizard

To run a Transient Cool analysis, you will need to model a mold. You can use the **Mold Surface Wizard** to create a mold around the part, feed system and cooling channels that you have modelled. Creating a mold surface is the first of three steps in creating a mold from within Autodesk Moldflow Insight. After you create the mold surface, the next step is to create a triangular surface mesh. The final step is to create the 3D mold mesh.

Creating a mold surface

Before you create your mold surface you must ensure that there are curves defining your *feed system* and *cooling channels*. Feed systems and cooling channels represented only by beam elements, without underlying curves, will not be considered.

1 Click **(Home tab > Create panel > Geometry)**.

2 Click **(Geometry tab > Create panel > Mold Surface)**, to open the Mold Surface Wizard.
You can accept the default dimensions or edit them as necessary.

NOTE: You can accept the default values and see how the mold looks. If it is not adequate, you can repeat step 2 and alter the dimensions until the mold is good.

3 Click **(Geometry tab > Selection panel > Properties)**, select **Curve** in the **By entity types** list, and confirm that all your feed system and cooling channels are defined by curves. If they are not, then you will need to fix this before the mold can be meshed.

4 Select **Beam** element from the **By entity types** list, then select **Tetrahedral** element and note which components are defined by these entity types.

NOTE: The mold dimensions must be such that all *tetrahedral* elements are fully enclosed within the mold. *Beam* elements may protrude.

5 If you identify any problems with the mold dimensions as a result of checking the properties, click **Mold Surface** again, and make the necessary alterations.

6 Click **Finish** to close the **Mold Surface Wizard**.
You are now ready to create the triangular surface mesh.

Preparing the mold mesh

Getting ready to generate a 3D mold mesh requires several steps, depending upon your starting point. In all cases, it is important to generate a mesh that is sufficiently fine on the internal surfaces adjacent to the part, feed system and cooling channels, yet sufficiently coarse on the outside edges to minimize the element count.

The mold internal edge lengths, that is the parts of the mold in contact with the part, the feed system and the cooling channels, should be sufficiently small that they define the shapes of the elements with which they are in contact.

CAD mold models To prepare a CAD mold model for generation of a 3D mold mesh, set an appropriate global edge length for all internal surfaces, including the part, feed system and cooling channel. Then set longer edge lengths on external mold surfaces to reduce the element count. The 3D mold mesh can then be generated in a single step.

STL mold models To prepare an STL mold model for generation of a 3D mold mesh, the first step is to generate a mold surface mesh. Once the surface mesh has been created, it should be checked for errors, then the mold external edge lengths should be increased relative to the internal edge lengths.

Mold models represented by regions and curves Mold models generated by the Mold Surface Wizard, or imported from Autodesk Inventor Tooling (AIT) are represented by regions and curves. To prepare the mold for generation of a 3D mold, the first step is to generate a mold surface mesh. Once the surface mesh has been created, it should be checked for errors, then the mold external edge lengths should be increased relative to the internal edge lengths.

Preparing the mold mesh

Changing mold internal and external edge lengths for CAD mold models

The part, feed system and cooling channels should all be meshed before you mesh the mold.

1 Make sure that the layers representing the part, feed system, cooling channels and mold are all visible.

2 Using the **Ctrl** key and the **Select** tool, rotate the mold and select all 6 faces.

3 Click (**Mesh tab > Mesh panel > Density**) to open the **Define Mesh Density** dialog.

4 Set the **Global edge length** to reflect the mold internal edge lengths, or accept the default value.

5 In the left-hand panel, select the first mold face, then deselect **Use global edge length** in the **Mesh density of selected entity** section. This will activate the **Target edge length** entry box.

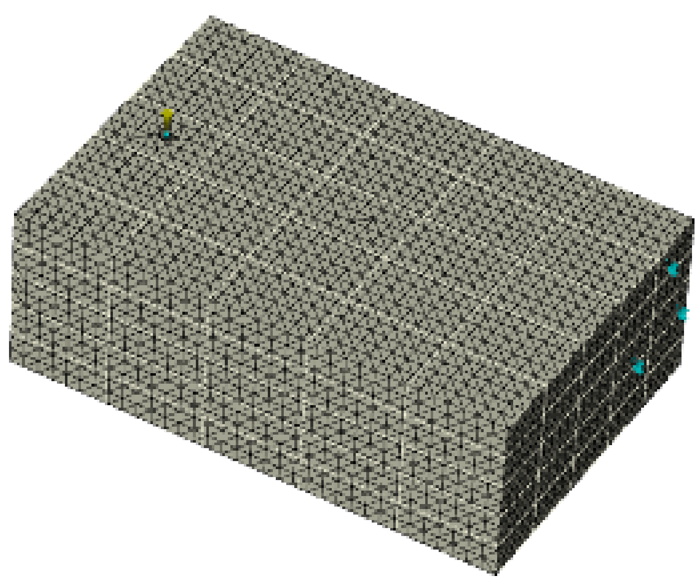
6 In the **Target edge length** box, enter a suitable edge length for the mold external edges. Mold external edges may be as much as 3-4 times as long as the internal edges.
7 Click **Apply** to accept the new external edge length for that mold face.
8 Repeat steps 5 - 7 for all the remaining mold faces, then click **Close**.

Generating the mold mesh

The mold mesh is prepared in a single step, or in two steps, depending upon the mold model.

Transient cool analyses are available only for 3D models, so the final mold mesh must be 3D. The 3D mold mesh can be created in a single step for CAD/STL molds, but you can create it in two steps if you would like to check the integrity of the mold surface mesh before generating the 3D mesh.

Molds generated by the Mold Surface Wizard, or imported from Autodesk Inventor Tooling (AIT) are composed of regions for the mold body and curves for the feed system and cooling channels. In these cases, the mold mesh is generated in two steps. A mold surface mesh is created first and it is advised that you check the mesh for errors, particularly around the feed system and cooling channel internal surfaces, and fix them before proceeding. The 3D mesh is generated in a second step.



CAD mold models	For imported CAD mold models, the mold can meshed in a single step.

STL mold models For imported STL molds, the mold can meshed in a single step, but to reduce the element count it is advisable to stop the meshing after the surface mesh has been generated in order to apply longer edge lengths to the mold external surfaces. The 3D mesh can then be generated.

Mold Surface Wizard molds A mold created by the Mold Surface Wizard is composed of regions for the mold body, and curves for the feed system and cooling channels. To generate a mold mesh for regions and curves, an initial mold surface mesh is created first. You should check the mold surface mesh for errors, paying particular attention to internal surfaces around the feed system and cooling channels. Once you are confident that the mold surface mesh is correct, you can generate the 3D mold mesh.

Molds imported from Autodesk Inventor Tooling (AIT) A mold created by the Mold Surface Wizard is composed of regions for the mold body, and curves for the feed system and cooling channels. To generate a mold mesh for regions and curves, an initial mold surface mesh is created first. You should check the mold surface mesh for errors, paying particular attention to internal surfaces around the feed system and cooling channels. Once you are confident that the mold surface mesh is correct, you can generate the 3D mold mesh.

Generating the mold mesh

Creating the mold mesh is the final step before running a Transient cool analysis.

Creating a mold surface mesh for regions and curves

1 Select the **Mold external surface** layers and make them active.

2 Click **(Mesh tab > Mesh panel > 3D Mold Mesh)** to open the **Generate 3D Mold Mesh** dialog.

3 Check the external and internal mold surface edge lengths on the **Surface** tab, and confirm that the **External mold surface edge length** is greater than the internal length. Make edits if necessary, but generally the defaults should suffice.

4 Click **Mesh Now**.
A dialog will appear, advising you to check the surface mesh once it has been created, and before you remesh to create the 3D mold mesh.

5 Click **OK** to close the message dialog and start the meshing process.

6 When the meshing is complete, check the mesh log for meshing error and fix them using the mesh repair tools.

You are now ready to create the 3D mold mesh.

Creating a 3D mold mesh for an imported CAD/STL mold

A 3D mesh is created for an imported CAD/STL model in a single step, after the part has been meshed.

Before you mesh the mold, make sure you have meshed the part. In the layers pane, uncheck the mold **Stl Representation**, click **(Mesh tab > Mesh panel > Generate Mesh)** to generate the part mesh, and use the **Mesh Repair** tools to optimize the part mesh, if necessary.

1 In the layers panel, check the mold layer to make it visible

2 Click on the mold to make it active, then click **(Mesh tab > Properties panel > Change).**

3 In the **Change Property Type To** dialog, select **Mold block (3D)** from the list.

4 Click **OK** to accept the selection and close the dialog.

5 Click **(Mesh tab > Mesh panel > 3D Mold Mesh)** to open the **Generate 3D Mold Mesh** dialog.

6 Edit the defaults in the **General** tab and the **Tetra** tab, if necessary.

NOTE: In general, the default values should be acceptable.

7 Click **Mesh Now** to start the 3D mesher.

8 Click **Close** to close the **Mesh Generation** dialog, if necessary.

Creating a 3D mold mesh for mold block surfaces defined by the Mold Surface Wizard

For mold block surfaces defined by the Mold Surface Wizard, creating the 3D mold mesh is the final step before running a Transient cool analysis.

Make sure you have created a *mold surface mesh* before you attempt to create a 3D mold mesh.

To convert a mold surface triangular mesh into a 3D mold mesh:

1 Double-click **Create 3D Mold Mesh** in the **Study Tasks** pane to open the **Generate 3D Mold Mesh** dialog.

2 Select the **Tetra** tab and check the values. Edit if necessary, but generally the defaults should suffice.

3 Click **Mesh Now** to generate the 3D mold mesh.

Generating the mold mesh

This dialog is used to generate a 3D mesh for the imported or created mold geometry.

To access this dialog, click **(Mesh tab > Mesh panel > 3D Mold Mesh).**

Generate 3D Mold Mesh tool

This panel is used to define and create a finite-element 3D mesh for a mold imported into, or created in Autodesk Moldflow Insight.

Not all tabs are visible all of the time. Different tabs will be visible depending on how you have generated your mold.

Surface tab (only when generating a surface mesh for a Mold Surface)	[illegible]

General tab (only when creating the 3D mold mesh)

Tetra tab

[illegible]

NOTE: Any mesh option marked with * can be saved as part of a workspace default.

Parallel solution method for Cool (FEM) analyses

Parallel solution technology is implemented as an option in the process settings for Cool (FEM) analyses to improve solution speed, especially for large models.

The parallel solution method is supported for shared memory multi-processor (SMP) systems, also known as multiple core systems. In SMP systems, all physical processors (cores) are in the same computer and access the full system memory, so data sharing is fast.

NOTE: Distributed memory clusters are not supported.

There are 4 options:

Automatic By default, the software automatically determines the most efficient number of threads to use based on CPU usage. This method takes advantage of available processing resources without overloading the machine. However, it may not always result in the fastest analysis due to a small overhead in reading the CPU usage, which may contribute to the overall solution time.

Single Thread (no parallelization) This means the parallel solution will not be used.

Maximum The analysis will be run using the maximum number of physical processors available for parallelization. This includes multiple cores, but does not include additional logical processors made available by enabling hyperthreading.

NOTE: The maximum number of threads used in the calculation is included in the Analysis log.

Specified number of threads

You can specify the number of threads you want to be used for parallelization.

NOTE: If the specified number of threads exceeds the number of physical processors (cores) available, the solver will default to using the maximum numbers of physical processors available.

If you have hyperthreading enabled, then the number of processors available will appear to be twice the number of physical processors. However, this does not result in the most efficient parallel execution. For best results, the number of threads specified for parallelization should not exceed the number of physical processors available on the system.

NOTE: For parallel analysis, the AMG matrix solver will always be used. The option to disable the AMG solver is only available for single-threaded analysis.

Parallel solution method for Cool (FEM) analyses

To access the parallel solution option, ensure that you have specified an analysis sequence that includes Cool (FEM).

Enabling the parallel solution

The **Number of threads for parallelization** option is set to **Automatic** by default. Other options include **Single thread (No parallelization)**, which means the parallel solution method will not be used, **Maximum**, or **Specify the number of threads** to be used for the parallel solution.

1 Click (**Home tab > Molding Process Setup panel > Process Settings**) to open the **Process Settings Wizard - Cool (FEM) Settings** dialog.
2 Click **Cool (FEM) Solver Parameters** to open the **Cool (FEM) Solver Parameters** dialog.
3 Change the **Number of threads for parallelization** option from **Automatic** to your preference.

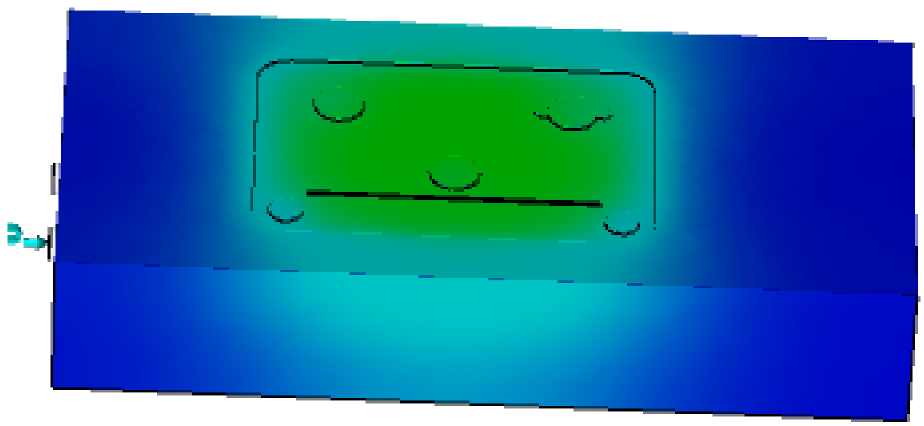
   NOTE: The number of threads specified should not exceed the number of physical processors available.

4 Click **Finish**.

Transient within cycle analysis

Transient cool analyses are used to simulate the change in mold temperature with time. Transient within cycle cool analyses simulate mold temperature deviations from the cycle average.

When an injection molding machine has been working continuously for a period of time, the mold temperatures eventually settle down to a consistent, cycle-averaged temperature. The In-cycle transient cool analysis can simulate the cyclical nature of the mold temperature, and enable you to see the deviations from the average temperature.



Transient within cycle analysis

The Transient Cool analyses use the finite element method to calculate the transient temperatures in the mold. To run a transient Cool analysis, you will need to select an analysis sequence that includes a Cool (FEM) analysis.

Running a Transient within cycle analysis

Run a Transient within cycle analysis to monitor the flow of heat in the plastic injection mold, during an injection cycle.

NOTE: If you imported a CAD/ STL mold first mesh the part, the feed system and cooling circuit, then mesh the mold.

NOTE: If you are modeling the mold in Autodesk Moldflow Insight, first mesh the part, the feed system and cooling circuit, then create a surface mesh for the mold, then create the 3D mesh for the mold.

1 *Import a model*.
2 Import or *model the feed system*, including the gate, runners and sprue.

NOTE: Make sure that the feed system is represented by curves, or it will not be considered in the mold.

3 *Import* or *model the cooling circuit*.

NOTE: Make sure that the cooling channels are represented by curves, or they will not be considered in the mold.

4 Click (**Home tab > Molding Process Setup panel > Analysis Sequence**) and select a Cool (FEM) sequence.

5 Click (**Home tab > Molding Process Setup panel > Process Settings**), select the *Selecting a Transient cool analysis* on page 24 analysis and also *Selecting a Transient cool analysis* on page 24 you would like to use.

6 Click **(Home tab > Create panel > Mesh)** to open the Mesh tab.

7 Click **(Mesh tab > Mesh panel > Generate Mesh)** and mesh the part, feed system and cooling system.

8 With only the meshed layer visible, click (**Home tab > Molding Process Setup panel > Injection Locations**) and set the injection location on the feed system.

9 With only the meshed layer visible, right-click **Create cooling circuits** in the **Study Tasks** pane and select **Set Coolant inlets** from the drop-down menu.

10 Place the coolant inlets in the appropriate places, edit the coolant properties, then close the **Set Coolant inlet** dialog box.

11 Confirm that all steps in the Study Tasks pane, up to **Create Mold Mesh**, have been completed and have a check mark associated with them.

12 *Importing a mold* on page 18, or click **(Geometry tab > Create panel > Mold Surface)** to *Mold Surface Wizard* on page 30.

13 If you created a mold surface using Autodesk Moldflow Insight, show the mold layer and click **(Mesh tab > Mesh panel > Generate Mesh)** to *Mold Surface Wizard* on page 30 for the mold.

14 With only the meshed layer visible, make the mold layer visible, click **(Mesh tab > Mesh panel > 3D Mold Mesh)** to *Generating the mold mesh* on page 33.

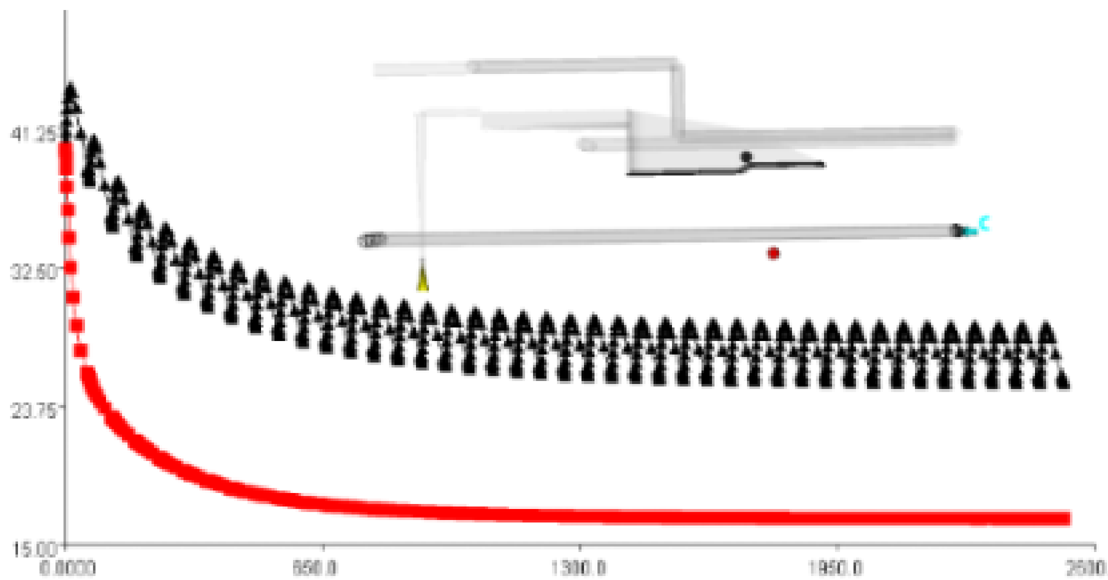
NOTE: Check the 3D mold mesh for errors and fix them before continuing.

15 Double-click **Start Analysis** in the **Study Tasks** pane to launch the analysis.

Transient from production start-up analysis

Transient cool analyses are used to simulate the change in mold temperature with time. Transient from production start-up cool analyses simulate the mold temperature variations from the initial production start-up cycle when the mold is cold, until the mold temperature stabilizes and reaches its optimum operating conditions.

When an injection molding machine is commissioned to manufacture a product, it may require many cycles before the mold reaches its optimum operating conditions. The Transient from production start-up cool analysis sequence will show the number of cycles required before the mold reaches these optimum operating conditions. In the example below, the black node on the mold is near the part and is affected by the in-cycle temperature variations. The red node is further from the part and is unaffected by in-cycle variations. It took 42 iterations for the mold temperature to reach steady-state.



NOTE: Transient cool analysis sequences are only available for 3D mesh types.

Transient from production start-up analysis

The Transient Cool analyses use the finite element method for calculating the results. To run a transient Cool analysis, you will need to select an analysis sequence that includes a Cool (FEM) analysis.

Running a Transient from production start-up analysis

Run a Transient from production start-up analysis to monitor the flow of heat in the plastic injection mold, from one cycle to the next to determine the number of cycles needed to reach a stable mold temperature.

NOTE: If you imported a CAD/ STL mold first mesh the part, the feed system and cooling circuit, then mesh the mold.

NOTE: If you are modeling the mold in Autodesk Moldflow Insight, first mesh the part, the feed system and cooling circuit, then create a surface mesh for the mold, then create the 3D mesh for the mold.

1. *Import a model*.
2. Import or *model the feed system*, including the gate, runners and sprue.

   NOTE: Make sure that the feed system is represented by curves, or it will not be considered in the mold.

   NOTE: If you import the feed system, make sure you set an injection location.

3. *Import* or *model the cooling circuit*.

   NOTE: Make sure that the cooling channels are represented by curves, or they will not be considered in the mold.

   NOTE: If you import the cooling channels, make sure you assign them the property **Channel**.

4. Click (**Home tab > Molding Process Setup panel > Analysis Sequence**) and select a Cool (FEM) sequence.
5. Select the *Selecting a Transient cool analysis* on page 24 analysis from the **Process Settings Wizard - Cool (FEM) Settings** dialog.
6. *Selecting a Transient cool analysis* on page 24 you would like to use from the **Cool (FEM) Solver Parameters** dialog
7. Confirm that all steps in the Study Tasks pane, up to **Create Mold Mesh**, have been completed and have a check mark associated with them, except 3D Mesh.
8. *Importing a mold* on page 18, or click **(Geometry tab > Create panel > Mold Surface)** to *Mold Surface Wizard* on page 30.

   NOTE: Make sure that all tetrahedral elements are fully enclosed by the mold surface.

9 Hide the mold layer and click **(Mesh tab > Mesh panel > Generate Mesh)** to mesh the part, feed system and cooling channels.

10 If you created a mold surface using Autodesk Moldflow Insight, show the mold layer and click **(Mesh tab > Mesh panel > Generate Mesh)** to *Mold Surface Wizard* on page 30 for the mold.

NOTE: Check the surface mesh for errors and fix any errors before continuing.

---

11 With the mold layer visible, click **(Mesh tab > Mesh panel > 3D Mold Mesh)** to *Generating the mold mesh* on page 33.

NOTE: Check the 3D mold mesh for errors and fix them before continuing.

---

12 Double-click **Start Analysis** in the **Study Tasks** pane to launch the analysis.