Autodesk® Moldflow® Insight 2012

AMI Result Plots

Autodesk®

Revision 1, 23 March 2012.

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

Contents

Result plots 1

Once an analysis finishes, a default set of results are listed for all analyses performed. You can then edit, rename, examine, overlay results, or even create new result plots.

Result plots

Use different tools to help you better analyse and understand the results of an analysis.

Examining a result plot

The Examine Result tool allows you to obtain precise result values in specific model areas.

To use the Examine Result tool, ensure you have a result displayed.

1 Click **Results tab > Plots panel > Examine**.

The mouse cursor will change to small cross-hairs.

2 Move the cursor to the location where you require an exact value and click the left mouse button once.

3 Right-click the mouse button once and choose **Select** to deactivate the Examine Result function.

NOTE: You can use the **Ctrl** key to examine more than one area at a time. Simply hold-down the **Ctrl** key and click on different areas on your model.

Overlaying result plots

The overlay function allows you to view more than one result at a time on the same model.

To overlay a result, you must have a result displayed.

Fill analysis results show how a part fills, and identifies any potential flow problems. You can overlay combinations of complementary fill analyses to gain a greater understanding of polymer flow characteristics. Typical combinations of analyses results can include the following:

Fill time and Air traps
Fill time and Weld lines

Fill time and Fibre orientation

TIP: To help visualize overlaid results, you can combine different plot display options. For example, you could display a pressure plot as a shaded result overlaid with a contoured Fill time result.

To overlay a result, follow the steps below.

1 In the **Study Tasks** pane, select a result by clicking the check box.
This is the displayed result.

2 Highlight another result, right-click and then select **Overlay**.
This is the highlighted result and is overlaid on the display result.

TIP: While overlaying plots you can set the transparency of a result to allow a display on different components, for example: cooling channels, and model. Highlight a result, right-click **Properties > Mesh Display**. Select the **Transparent** option button and set the **Opacity** between the range from 0 to 1, where 0 means that the model is invisible.

---

3 Right-click the original result and select **Activate** to make it the active result. When two results are overlaid, only the active result can be animated.

NOTE: It is a good idea to overlay a result onto a line contour plot. This way it is very easy to view and interpret both plots on the model.

Viewing intermediate results (Midplane or Dual Domain)

Viewing intermediate results while a result is running allows you to retrieve information as they are produced.

During a Fill+Pack analysis, intermediate results are saved at specified intervals. Intermediate results offer additional time-animation capabilities and recovery options.

1 Click  **Home tab > Molding Process Setup panel > Process Settings**.

2 On the **Fill** or **Fill+Pack Settings** page of the Wizard, click **Advanced Options**.
The **Fill+Pack Analysis Advanced Options** dialog appears.

3 In the **Solver Parameters** area, click **Edit**.

4 Click the **Intermediate Output** tab and select **Yes** from the **Dynamically update result display analysis** drop-down list.

5 Click **OK** twice and complete the **Process Settings Wizard**.

6 Click  **Home tab > Analysis panel > Start Analysis** to launch the analysis.

As soon as the first set of intermediate results are available, the available plots are listed under the **Results** node in the **Study Tasks** pane. You can view these plots or add new plots in the same way as for completed analyses.

Renaming result plots

You can change the name of a result plot from the default to more accurately describe the plot. This is particularly useful when you have several plots of the same result, each with different plot properties.

There are two ways to rename a plot:

- Right-click on the plot and select **Rename**.
- Click on the plot to select it, then press **F2** on your keyboard.

New plot 2

You can create new result plots in addition to the default set.

Creating new plots allows you to duplicate and customize the plot and its properties without affecting the default set.

New plot

You can create a new result plot to include results in addition to the default results.

Creating a new result plot

You can create a new result plot to include results other than the default results.

Once an analysis finishes, a default set of results are listed for all analyses performed. You can create new results at the completion of an analysis and display it in a separate window if required.

NOTE: To specify the default results created during an analysis, go to **File > Options**.

---

1. Click **Results tab > Plots panel > New Plot > Plots**, or right-click **Results** in the **Study Tasks** pane and select **New Plot**.

   The **Create New Plot** dialog appears.
2. In the **Available results** list, click on the plot to be created.
3. In the **Plot type** area, select the type of plot to be created.

   You can create an **Animation plot**, **XY plot**, **Path plot**, **3D slice plot**, **Highlight plot**, **2D slice plot**, **Probe XY plot**, **Probe plot**, or **Slice on mold**.

   NOTE: The available plot types depend on the mesh type and the plot you selected in Step 2.

   ---
4. Select the **Open a new window** check box if you want the result to be created in a separate window.
5. Click the **Plot Properties** tab and make any required changes to the result settings.

The options that appear depend on the selected plot and plot type.

6 Click **OK** to close the dialog and display the new result.

Plotting a result along a path through the model

You can display several results using the **Path Plot** plot type.

The **Path Plot** plot type is an XY plot with the result on the Y axis and the coordinate, or distance from path origin, of the locations that you select on the model. This enables you to plot, for example, pressure as a function of distance from the gate, or part deflection values along an edge of the part.

1 Click **Results tab > Plots panel > New Plot > Plots**.

2 Select the required plot in the **Available results** list.

3 Select **Path plot** in the **Plot type** area.

4 In the **Plot Properties** tab, select the **Path Plot** tab.

5 In the **X Axis** drop-down list, select the variable you want to display on the X axis of the plot.

As you select points along a path through the part, the X axis can display either the net distance from the first location you selected (the path origin), the cumulative distance of each of the line segments joining the locations, or the X, Y or Z coordinate of each selected location. Note that the coordinates will be on the corresponding global axis unless you have defined and activated a local coordinate system.

NOTE: For ease of plotting the path on the model, it is best to display the model's element lines. Click on the **Mesh Display** tab and select **Element lines** from the **Edge display on undeformed part** box.

---

6 Click **OK** to create the new plot.

You will see an empty XY graph with the part visible behind it.

7 Click on the part to specify the start location of where you want to trace through the part. Continue to click on the part along the desired path.

As you select each location, the selected node is displayed in the **Entity IDs** dialog and the XY graph is updated with the result on the Y axis and your selected variable on the X axis.

8 To edit or clear the XY graph and start again, select and delete the node numbers list in the **Entity IDs** dialog and press **Return**.

9 Repeat the previous step to create more curves on the XY graph.

NOTE: If you activate a different tool while working with the probe XY plot, you need to reselect **Results tab > Plots panel > Add XY Curve** to create new curves.

Plotting a result on a slice through the mold (3D models)

Results generated by a 3D analysis on the mold block can be displayed using the **Slice on mold** plot type. This activates a cutting plane on the model result display. The cutting plane allows you to view results through the thickness of the mold block, that is, to *see inside* the model.

To generate a result on the mold block, you must include the mold block in your analysis model.

Currently, the only result that is calculated on mold block elements is the **Mold internal temperature** result of 3D Cool analysis. The slice on mold plot type is the default plot type for this result.

1. Click  (**Results > New Plot**).
2. Select the **Mold internal temperature** result in the **Available results** list.
3. Select **Slice on mold** in the **Plot type** area.

   A cutting plane appears through the mold block in the model display window.
4. Use the cutting plane tools in the **Viewer** toolbar to change the orientation of the cutting plane or move it through the model.

NOTE: If you activate a different tool while working with the slice on mold plot, you need to click  **Edit Cutting Plane** or  **Move Cutting Plane** in the **Viewer** toolbar to change the orientation or location of the slice on mold plot.

Plotting a result on a plane through the part (3D models)

Several results created from a 3D analysis can be displayed using the **Probe plot** plot type, which allows you to specify multiple result planes in the part. You need to specify the direction and location of the planes.

1. Click **Results tab > Plots panel > New Plot > Plots**.
2. Select the desired result in the **Available results** list.
3. Select **Probe plot** in the **Plot type** area, then click **OK.**

   The **Probe Plane** dialog opens.
4. Specify the **Plane normal (dx, dy, dz)**.

   - The default plane normal is 1.0 0 0, which sets the plane normal (perpendicular) to the X axis, in the YZ plane.
   - A plane normal of 0 1 0 sets the plane normal to the Y axis, in the XZ plane.
   - A plane normal of 0 0 1 sets the plane normal to the Z axis, in the XY plane.
   - A plane normal of 1 1 0 sets the plane normal at a 45-degree angle.

5 Specify the **Plane point (x, y, z)** in one of two ways:
   a Click on the model.
   b Enter the x y z coordinates in the **Plane point** text box, and click **Apply**.
6 The result will be displayed on the plane that cuts through the part.
7 Repeat steps 4 and 5 to specify multiple planes.
8 Click **Reset** to remove all specified probe planes from the display.

NOTE: If you activate a different tool while working with the probe plot, you need to double-click on the **[result name]:Probe plot** plot name in the **Results** list to open the **Probe Plane** dialog again.

Plotting a result through the part thickness (3D models)

Several results created from a 3D analysis can be displayed using the **Probe XY plot** plot type.

A Probe XY plot displays the result on the Y axis and distance through the part on the X axis.

1 Click **Results tab > Plots panel > New Plot > Plots**.
2 Select the desired result in the **Available results** list.
3 Select **Probe XY plot** in the **Plot type** area.
4 Select the **Plot Properties** tab and ensure that **Overlay with mesh** is selected.
5 Click **OK** to create the new plot.
   You will see an XY graph with the part visible behind it.
6 Click on a location on the surface of the part.
   Autodesk Moldflow Insight will automatically locate the opposite side of the part, trace a path from the selected location to the opposite side, and then plot the result against distance along this path.
7 Repeat the previous step to create more curves on the XY graph.

NOTE: If you activate a different tool while working with the probe XY plot, you need to reselect **Results tab > Plots panel > Add XY Curve** to create new curves.

Displaying profiled intermediate results as an XY plot

Profiled intermediate results allow you to view a variable, for example temperature or velocity, across the cavity thickness at selected locations in the part and at various times from the start of injection.

By default, profiled intermediate results are only displayed as a shaded plot, so if you wish to view them as an XY plot you need to create them manually.

NOTE: The number of intermediate results sets to be written during filling and packing can be set in the advanced options of the Fill+Pack analysis settings.

---

1. Right-click the Fill+Pack result folder in the **Study Tasks** pane and select **New Plot**.
2. In the **Available results** list, select a result type for which profiled intermediate results exist (for example; temperature, velocity) and select **XY Plot** from the **Plot Type** options.
3. In the **Plot Properties** tab, select the **XY Plot Properties (1)** tab.
4. Select **Normalized thickness** from the **Independent Variable** drop-drown list.
5. Select the **Time** from start of injection at which you want to examine the profile data.
6. Click **OK** to create the new plot.
7. Click on the required locations in the part to display a 2D graph of the profiled result at each respective location.
8. To view profiled intermediate result at a different time during injection, either create a new plot using the procedure described above, or edit the properties of the plot you have created by right-clicking the plot name and selecting **Properties**.

Adding a curve to a 2D plot

You can add a curve to an existing plot to customize it.

If you create a global XY plot, which is a result calculated across the whole part, such as clamp force, all you need to do is display the result to view the curve. For mesh-dependent XY plots, such as pressure or temperature vs. time, you need to create a new result, and then add curves to the plot using the method described below.

1. Click **Results tab > Plots panel > New Plot > Plots**.
   The **Create New Plot** dialog appears.
2. Select the required plot from the **Available results** section.
3. Select the **XY plot** option in the **Plot type** section.
4. Click the **Plot Properties** tab and ensure that **Overlay with mesh** is selected.
5. Click **OK**.
   The **Create New Plot** dialog closes and the **Entity IDs** dialog is active and waiting for input from the model pane.
6. Click on the part in the model pane to add a new curve to the XY plot. You may add up to five new curves to the plot.

NOTE: Click **Select** to close the **Entity IDs** dialog in order to prevent accidentlly selecting further entities to plot.

New plot

This dialog is used to create a new customized plot or edit various properties from the available analysis results.

To access this dialog, click (**Results tab > Plots panel > New Plot > Plots**).

Create New Plot dialog—Result Selection tab

This dialog is used to create a new customized plot from the available analysis results.

Accessed this dialog by clicking (**Results tab > Plots panel > New Plot > Plots**) or by right-clicking the **Results** section in the **Study Tasks** pane and selecting **New Plot**.

The **Result Selection** tab of this dialog is used to select the result to be created, including the type of result, the type of plot to create the result in, and whether you want to output the result into a separate window. Creating a result in a new window can be useful when comparing results.

Create New Plot dialog—Plot Properties tab

This dialog is used to view and edit various properties for the displayed result.

The dialog can be accessed by clicking (**Results tab > Properties panel > Plot Properties**), or by right-clicking on the result in the **Study Tasks** pane and selecting **Properties**.

NOTE: Only tabs relevant to the selected result will be displayed.

Custom plot 3

Using the Create Custom Plot tool allows you to further customize result plots.

Custom plot

Creating custom plots allow you to adjust the plots based on your specifications.

Creating a custom plot—birefringence

The default birefringence results show stress-optical effects for light coming from one of two global axial directions (+Z, -Z) and for light of a fixed wavelength (546.1 nm).

If these choices are inadequate (for instance, your model is oriented so that the important optical direction is a different axis), you can create a custom plot showing results for light originating from any other direction and of any wavelength.

NOTE: This plot is available only if you have selected the **Birefringence analysis if material data includes optical properties** checkbox in the Fill+Pack page of the process settings wizard and the material has measured optical properties.

---

1. Define a local coordinate system so that any one of the three cardinal axes (X, Y, or Z) is pointing *towards* the direction *from* which the light is coming.
2. Click (**Results tab > Plots panel > New Plot**) and select **Custom** from the drop down list.
3. Select **Birefringence** from the **Create Custom Plot** pane.
4. Enter a name for the plot in the **Plot Name** text box.
5. Select the local coordinate system and the axis which points towards the light source.
6. Choose one of the three plot types **Phase shift**, **Retardation** or **Retardance tensor**. For **Phase shift**, enter a wavelength.
7. Click **OK**.

The plot is displayed and the plot name is listed in the **Study Tasks** pane under **User-Defined Plots**.

Rotating the model to match the global coordinate system

The default parting plane for a mold is the XY plane of the global coordinate system. If your part is modeled with a different parting plane, you can rotate the model so the global coordinate system's XY plane coincides with the parting plane.

The model must be rotated before you run the analysis.

1. Click (**Geometry tab > Utilities panel > Move**) and select **3 Points Rotate** from the drop down list.
2. Select all the elements in the model, either with the mouse or (**Geometry tab > Selection panel > Select All**).
3. Select three points on the model, all of which are on the parting plane. The points must not be in a straight line.
4. Click **Apply**.

The part is now rotated as required.

Specifying a local coordinate system for the parting plane

The default parting plane for a mold is the XY plane of the global coordinate system. If your part is modeled with a different parting plane, you can specify a local coordinate system for the parting plane.

1. Click (**Geometry tab > Local Coordinate System panel > Create LCS**).
2. Select three points on the model, all of which are on the parting plane. The points must not be in a straight line.
3. Click **Apply**.
4. Click (**Results tab > Plots panel > New Plot > Custom**).
5. Select **Clamp Force On Parting Plane**.
6. Select the local coordinate system you created, and set the parting plane normal to the Z axis.
7. Click **OK**.

   The new plot appears in the results list as a user-defined plot.

Creating a custom weld line angle result

Specifying a custom weld line angle allows you to see the effect of the meeting angle on the length and severity of the weld line.

The **Weld Lines** custom plot is available for Midplane and Dual Domain studies. It allows you to specify the angle at which the flow fronts meet and create a weld line.

1. Click (**Results tab > Plots panel > New Plot > Custom**).

2 Select **Weld Lines** from the **Create Custom Plot** pane.

3 Enter a name for the plot in the **Plot Name** text box.

4 Enter a meeting angle into the **Max. meeting angle** text box.

The default meeting angle is 135 degrees.

5 Click **OK**.

The plot is displayed and the plot name is listed in the **Study Tasks** pane under **User-Defined Plots**.

You can create several plots with different meeting angles to compare the effects. Click the check box next to each plot name to display each plot in turn and compare the effects of the meeting angle.

TIP: To assist in comparing plots, make the color of the weld lines different in each plot. You may also find it useful to overlay plots to compare the weld lines.

Custom plot

This dialog is used to customize plots that are not possible to create using the Calculated Plots functionality in the Results menu.

The dialog can be accessed by clicking  (**Results tab > Plots panel > New Plot > Custom**).

NOTE: Plots listed in the dialog depend on the results available for the current study.

Create Custom Plot—Birefringence

This dialog, which can be accessed by clicking  (**Results tab > Plots panel > New Plot > Custom**), offers a selection of customized plots that are not possible to create using the Calculated Plots functionality.

NOTE: The custom plots listed in the dialog depend on the results available for the current study.

The **Birefringence** plot, which requires results from a Fill+Pack analysis and a Warp analysis, displays stress-optical (birefringence) results for a custom direction (from a positive axis of a local coordinate system).

Phase Shift	The phase difference between horizontally-polarized and vertically-polarized light, expressed in degrees. 180 degrees is half of one wavelength.

Retardation	The phase difference between horizontally-polarized and vertically-polarized light, expressed in nanometers.
Retardance tensor	The integral of the change in refractive index tensor along the light path.

Create Custom Plot—Clamp Force on Parting Plane

This dialog, which can be accessed by clicking  (**Results tab > Plots panel > New Plot > Custom**), offers a selection of customized plots that are not possible to create using the Calculated Plots functionality.

NOTE: The custom plots listed in the dialog depend on the results available for the current study.

The **Clamp Force on Parting Plane** plot, which requires results from a Fill/Fill+Pack analysis, is used to create an XY plot of clamp force vs time based on a specified clamp-open direction. The default Clamp force result created by the filling analysis assumes a parting plane normal to the global Z direction. This custom plot therefore removes the requirement to orient the part in a specific direction prior to analysis.

Create Custom Plot—Deflection plot

This dialog, which can be accessed by clicking  (**Results tab > Plots panel > New Plot > Custom**), offers a selection of customized plots that are not possible to create using the Calculated Plots functionality in the Results menu.

NOTE: The custom plots listed on the dialog depend on the available results for the current study.

The **Deflection plot** custom plot allows you to create customized plots showing shrinkage and warpage based on results from a Warp analysis:

DEFL	Displays shrinkage and warpage with a specified magnification scale factor.
DEFL C	Displays shrinkage and warpage with a specified magnification scale factor, centering the display on the node(s) specified.
DEFL W	Displays warpage without shrinkage with a specified magnification scale factor, based on the position of two specified nodes.
DEFL W2	Displays warpage with uniform shrinkage, based on a specified shrinkage value.

Create Custom Plot—Modulus

This dialog, which can be accessed by clicking (**Results tab > Plots panel > New Plot > Custom**), offers a selection of customized plots that are not possible to create using the Calculated Plots functionality.

NOTE: The custom plots listed on the dialog depend on the available results for the current study.

The **Modulus** plot requires results from a 3D Fiber orientation analysis, and displays the elastic modulus in the specified principal direction, E11, E22 or E33, based on the global coordinate system or a specified local coordinate system.

Create Custom Plot—Weld Lines

This dialog, which can be accessed by clicking (**Results tab > Plots panel > New Plot > Custom**), offers a selection of customized plots that are not possible to create using the Calculated Plots functionality.

NOTE: The custom plots listed on the dialog depend on the available results for the current study.

The **Weld Lines** custom plot allows you to specify a meeting angle. This allows you to investigate the effect of the meeting angle on the length of the weld lines.

Calculated plot 4

You can create new mathematically calculated plots.

Calculated plot

Perform plot calculations to further analyze results.

Creating a new result by performing a calculation on existing results

You can perform a calculation on existing results to analyze between two results.

Mathematical functions are used to calculate the results.

1 Click **Results tab > Plots panel > Calculated**.
The **Create Calculated Plot** dialog appears.

2 In the **New plot** text box, enter a name for the plot you are about to create.

3 Click (**Browse**) next to the **Result** text box in the **Data A** area, and select the first result to be used.
You can also select a particular time or thickness value from the drop-down list if they are available.

4 In the **Function** drop-down list above, select the mathematical function that you want to perform on the selected result.

5 In the **Operator** drop-down list, select the required mathematical operator in order to either; add result B to result A, subtract result B from result A, multiply result A by result B, divide result A by result B.

6 Perform steps 2-5 above for the second result in the **Data B** area.

NOTE: The second data item can also be a constant value. Select **(Constant)** in the **Function** drop-down list and then specify the constant value in the text box that appears.

---

7 Click **Apply**, or **OK** to calculate and display the new plot.
You will find the new plot listed in a **User-Defined Plots** folder at the bottom of the **Study Tasks** pane.

Calculated plot

This dialog is used to create a new mathematically calculated plot.

To access this dialog, click (**Results tab > Plots panel > New Plot > Calculated**).

Create Calculated Plot dialog

This dialog is used to create a new plot by performing a mathematical calculation on existing results.

This dialog can be accessed by clicking **(> Results tab > Plots panel > New Plot > Calculated**).

Select Result dialog

This dialog, which is accessed by clicking (**Results tab > Plots panel > New Plot > Calculated**) and then (**Browse**) in the **Create Calculated Plot** dialog. This dialog is used to select an existing result, or previously created calculated plot, as a data item to be used in a plot calculation.

Plot calculation functions

This topic provides you with a list of the mathematical functions that can be used when creating a new user-defined plot in Autodesk Moldflow Insight, and a description about the action that the function performs on the existing plot.

Function	Description
Sin	Calculates the sine of the expression (in radians).
Cos	Calculates the cosine of the expression (in radians).
Tan	Calculates the tangent of the expression (in radians).
Log	Calculates the logarithm of the expression (base “e”).
Log10	Calculates the logarithm of the expression (base “10”).
Exp	Exponentiate the expression (base “e”).
Abs	Calculates the absolute value of the expression.
Sqrt	Calculates the square root of the expression.
Sign	Evaluates to 0, 1 or -1 if the expression evaluates to 0, >0 or <0 respectively.
Nint	Calculates the nearest integer to the expression.

Function	Description
Constant	Uses a constant value for the expression.

To convert from degrees to radians, multiply by pi/180 (about 0.01745) before applying Sin, Cos or Tan.

Plot properties 5

Plot properties enables you to customize animation settings, result scale, and part display, and select from several result display methods. You can also use the Plot properties feature to specify the property settings for all result types.

Plot properties

The Plot properties dialog is used to view and edit result displays and specify the property settings for all result types.

Editing plot properties

After you have created a result, you may need to edit or make changes to its settings.

For example, you may have created an animated fill time result using 16 frames, and later want more frames included in the result in order to show the required detail.

1. Select a result, and then click **Results tab > Properties panel > Plot Properties**, or right-click the result in the **Study Tasks** pane and select **Properties**.
2. Click on the tabs and make the required change(s) to the result settings.
3. Click **OK**.

   The result displays with the new settings.

Adding notes to results

NOTE: This feature is available only for analyses that produce a results plot.

---

1. Select the result to which you would like to add a note.
2. Click **Results tab > Plots panel > Plot Notes...**.
3. Enter the text in the space provided.
4. Click **OK**.

NOTE: Each Plot Note is associated to one single result, and consequently does not appear in other results.

Plot properties

The Plot properties dialog is used to view and edit various properties of displayed results.

The dialog can be accessed by clicking (**Results tab > Properties panel > Plot Properties**) or by right-clicking on the result in the **Study Tasks** pane and selecting **Properties**.

NOTE: Only tabs relevant to the selected result will be displayed.

Plot properties dialog—General Tab

The **General** tab of this dialog is used to change the formatting of the current material property plot, in particular the scaling of the X and Y axes and the size and location of the plot legend.

To access this dialog, right-click on the Material Selection icon in the **Study Tasks** pane and select **Details**. From the dialog that appears, select either the Rheological Properties tab or the pvT Properties tab, depending upon the plot you are interested in. Click the relevant **Plot** button, then click

 **Plot Properties** in the bottom left of the dialog.

NOTE: Only tabs relevant to the current type of material data plot will be displayed.

Plot properties dialog—Methods tab

Specify how the result is displayed on the model.

You can display the result shaded or as line contours across the model. Line contours plots are the preferred option for accurate interpretation of results. You can also specify how to display the following types of results:

- vector (Orientation at skin)
- tensor

Plot properties dialog—Animation tab

Set the animation properties of the displayed result, such as the number of frames, or whether each frame displays a cumulative result.

NOTE: The available animation options also depend on the intermediate results output set as advanced options before running the analysis.

Single dataset Animates the result values through the part at a specific time during the molding cycle. In other words, there is

	only one set of data used in the animation and the values within that dataset are animated.
Multiple dataset	Creates an animation consisting of multiple sets of data at various values of a specified independent variable, generally Time or Normalized thickness.

You can set the number of frames used in the animation. For example, increase the number of frames to smooth an animation and see the material flow progress in smaller steps. This can be useful when investigating material flow through complex geometry.

Plot properties dialog—Scaling tab

Specify how the minimum and maximum values on the result plot are to be selected for an animated result.

NOTE: Results are scaled to the visible layers unless you specify minimum and maximum values.

Plot properties dialog—Part Display tab

Specify how edges are displayed on deformed and undeformed parts, and also the surface opacity of the parts.

Plot properties dialog—Optional Settings tab

Specify how colors are displayed when viewing a result.

Plot properties dialog—2D Slice Plot tab

Set the parameter that will be held constant in a 2D slice plot when viewing a molding window result.

Plot properties dialog—Highlight tab

Change the highlight properties of the result plot.

Depending on the result selected, you can change either the line color or the overlay result displayed on the plot. These could be, for example, an Air traps or Weld lines plot.

Plot properties dialog—Deflection tab

Configure how warpage results are displayed.

Plot properties dialog—Path Plot tab

The **Path Plot** tab of the **Plot properties** dialog is used to create an XY plot that shows the result property with respect to model geometry.

For example, if you want to plot the change in deflection along the bottom edge of a part, and deflection is primarily in the X direction, select X component deflection and pick nodes along the bottom edge. The node that was selected first becomes the reference point.

Plot properties dialog—XY Plot Properties (1) tab

Specify formating options pertaing to the selected XY plot.

You can specify the X-axis variable, whether you want data points, legend or mesh displayed, legend positioning, and you can add or delete curves.

NOTE: In order to quickly add curves to a plot, ensure that **Overlay with mesh** is selected.

Plot properties dialog—XY Plot Properties (2) tab

Specify the range of values and titles to be displayed on the XY plot.

Plot properties dialog—Mesh Display tab

The **Mesh Display** tab of the **Plot properties** dialog is used to specify how the mesh is displayed on the model when viewing the result. You can specify how the mesh edge is displayed, and whether part filling is solid or transparent. You can also specify whether the mesh is displayed on the deformed, undeformed, or all areas of your model.

Plot properties dialog—Tensor tab

The **Tensor** tab of the **Plot properties** dialog is used to select the tensor component, which will configure the colors that fiber orientation tensor results are displayed in. The tensor has 9 components, T_{ij} (i, j = 1, 2, 3); however, the tensor is symmetric so only 6 are available.

TIP: Tensors can be displayed as axes or ellipses from the **Methods** tab.

Plot properties dialog—Vector tab

Specify how both the scale and magnitude of elements are visually represented when displaying a vector result.

Examples of vector results include velocity results and material orientation results.

If a vector result is shaded, the Color may represent the magnitude of the vector, or the value of the X, Y, or Z component. If vector is plotted as a dart, the dart color is determined in the same manner as the shaded plot, but its length may be fixed as a constant (average mesh element size), or change according to the vector's magnitude (vector norm).

Plot Properties dialog—Viscosity Plot tab

The **Viscosity Plot** tab of this dialog is used to specify the temperature values for the curves to be displayed on the viscosity plot, and the pressure value upon which the viscosity values are based . This option is particularly useful when comparing the viscosity properties of two materials.

To access this dialog, right-click on the Material Selection icon in the **Study Tasks** pane and select **Details**. From the dialog that appears, select the Rheological Properties tab . Click the **Plot Viscosity** button, then click  **Plot Properties** in the bottom left of the dialog.

NOTE: Only tabs relevant to the current type of material data plot will be displayed.

NOTE: The pressure setting will have no effect on the viscosity plot if there is no pressure dependence in the viscosity data. This will be the case if the Second order viscosity model is selected, or the D3 term of the Cross-WLF Viscosity Model is equal to zero.

Global plot properties 6

You can apply modify plot properties and apply the changes globally.

Global plot properties

Modify the plot properties and save the changes globally.

Saving modified plot settings

If you have made a change to the properties of one of the results listed in the **Study Tasks** pane, you can apply the change to all results if required.

TIP: You can click  **> Options** to set the default results that are created during an analysis .

---

1 Select a result from the **Study Tasks** pane and change one of its properties.
(For example: right-click the result, select **Properties**, and change an **Element Edge Display** option in the **Mesh Display** tab).

2 Click **Results tab > Properties panel > Save Defaults > Save Global Plot Properties**.
The **Save Global Plot Settings** dialog appears.

3 In the **Modified plot settings** list, select the plot settings that you want to have saved globally.

4 Select the **Update all plots with the new settings** check box, and then click **Yes**.

TIP: Alternatively, right-click on the result node in the **Study Tasks** pane and select **Properties**. You can then manually change the properties.

TIP: You can use **Results tab > Properties panel > Save Defaults > Save Properties of Current Result** to specify that all results that are created hence are of the same format as the current plot. For example, you may want some Stress analysis results to be created as an XY plot by default.

The following settings can be saved globally:

- Optional Settings: Color group
- Optional Settings: Cut with Capping

- Optional Settings: Gouraud shading
- Methods: Contour group
- Mesh Display: Element Surface Display group
- Mesh Display: Element Edge Display groups

Global plot properties

You can save global plot settings for result plots.

To access this dialog, click  (**Results tab > Properties panel > Save Defaults > Save Global Plot Properties**).

Save Global Plot Settings dialog

This dialog is used to save global plot settings for result plots. Once you have a result plot open, you can apply modified plot settings to update all plots, or just for the plot you have active. To access this dialog, click  (**Results tab > Properties panel > Save Defaults > Save Global Plot Properties**).

Plot data 7

You can save your results in a variety of formats, including XML, Patran and ASCII (*.txt), from the **Results** toolbar.

You can also export warpage mesh and geometries predicted from a Warp analysis to an STL or UDM format.

Plot data

Saving plot data to different formats allow you to save various facets of the results to share and view on different platforms.

Saving XY plot data to a text file

You can export XY plot data to an ASCII file (*.txt) at the completion of an analysis. This ASCII file is viewable in a spreadsheet program.

NOTE: The set of default results created during an analysis can be set in **Options**.

---

1. Choose the relevant XY plot from the **Results** plot list.
2. Click **Results tab > Export and Publish panel > XY Plot**.
   The **Publish** dialog appears.
3. Enter a File name and locate the destination folder of your choice.
4. Click **Save**.

Saving plot results in XML format

It is possible to export Autodesk Moldflow Insight result data in XML (*.xml) format.

NOTE: The set of default results created during an analysis can be set in **Options**.

---

1. Choose the relevant XY plot from the **Results** plot list.
2. Click **Results tab > Export and Publish panel > XML**.
   The **Publish** dialog appears.
3. Enter a File name and locate the destination folder of your choice.
4. Click **Save**.

Saving plot results in Patran format

It is possible to export Autodesk Moldflow Insight element results data in Patran (*.nod) format.

NOTE: The set of default results created during an analysis can be set in **Options**.

---

1 Choose the relevant XY plot from the **Results** plot list.

2 Click **Results tab > Export and Publish panel > Patran**.
The **Patran** dialog appears.

3 Enter a File name and locate the destination folder of your choice.

4 Click **Save**.

Saving plot results in XML format

Results can be saved as an XML text file for interfacing with other software programs, such as ANSYS.

To save the plot results in XML format, you first need to have the result displayed.

XML plot results file structure

In the exported XML file, data is separatde into blocks. Each block has an index number and its contents depend on the data type (mesh data on elements or nodes; non-mesh data; and highlight data).

For mesh data, each block:

- Corresponds to one animation frame.
- Contains independent variable names, values and units.
- Contains dependent variable name, values and unit.
- Contains data type element or node ID and data values.
- Contains layer information. Included only if the data has layer information.

Non-mesh data is structured similarly to mesh data but has no element or node IDs.

Highlight data has only one block with one or more groups, each containing a set of point coordinates.

XML plot results file example explained

The following is an extract of an XML plot results file. Explanations are displayed in **bold** in comments after the // characters and are not part of the file.

```
////////////////////////////////////////////
//Exported mesh data:
////////////////////////////////////////////

<?xml version="1.0"?>

<MoldflowXMLDocument>

<!--
*****************************************************************
 -->

<!--
*****************************************************************
 -->

<!--File Name: C:\temp\stress.xml--> //file name

<!--
*****************************************************************
 -->

<!--
*****************************************************************
 -->

<HEADER>

<NAME> Moldflow Simulation Results XML Writer</NAME>

<Version> 1.00</Version>

</HEADER>

<Dataset Name="Stress, Mises-Hencky (stress)" ID="6604">
//dataset name

<DataType> ELDT(Element data)</DataType> //data type: mesh
 (element or node), highlight, or non-mesh data

<DeptVar Name="Stress, Mises-Hencky (stress)" Unit="Pa"/>
 //dependent variable name

<NumberOfComponents> 1</NumberOfComponents>

<NumberOfIndpVariables> 2</NumberOfIndpVariables>

<IndpVar Name="Load factor" Unit="%"/> //active independent
 variable name

<IndpVar Name="Normalized thickness" Unit=""/> //active
independent variable unit name

<Blocks>

<NumberOfBlocks> 2</NumberOfBlocks>
```

```
<Block Index="1">

<IndpVar Name="Load factor" Value="100.000000" Unit="%"/>
 //first active independent variable name and value

<IndpVar Name="Normalized thickness" Value="-1.000000"
Unit=""/> //second active independent variable name and
value

<NumberOfDependentVariables>
460</NumberOfDependentVariables>

<Data>

<ElementData ID="1"> //data type (element or node) id

<DeptValues> 2.2634e+006</DeptValues> //data values (one
value for scale and three or six for vector/tensor)

...

...

 </Data>

</Block>

</Blocks>

</Dataset>

</MoldflowXMLDocument>

////////////////////////////////////////////////

//Exported non-mesh data:

////////////////////////////////////////////////

<?xml version="1.0"?>

<MoldflowXMLDocument>

<!-- ********************************************** -->

<!-- ********************************************** -->

<!--File Name: C:\temp\xyplot.xml-->

<!-- ********************************************** -->

<!-- ********************************************** -->

<HEADER>
```

```
<NAME> Moldflow Simulation Results XML Writer</NAME>
<Version> 1.00</Version>
</HEADER>
<Dataset Name="% Shot weight" ID="1160">
<DataType> NMDT(Non-mesh data)</DataType>
<DeptVar Name="% Shot weight" Unit="%"/>
<NumberOfComponents> 1</NumberOfComponents>
<NumberOfIndpVariables> 1</NumberOfIndpVariables>
<IndpVar Name="Time" Unit="s"/>
<Blocks>
<NumberOfBlocks> 39</NumberOfBlocks>
<Block Index="1">
<IndpVar Name="Time" Value="0.171670" Unit="s"/>
<NumberOfDependentVariables> 1</NumberOfDependentVariables>
<DeptValues> 4.9860e+000 </DeptValues>
</Block>
...
...
</Blocks>
</Dataset>
</MoldflowXMLDocument>
////////////////////////////////////////////////
//Exported highlight data:
////////////////////////////////////////////////
<?xml version="1.0"?>
<MoldflowXMLDocument>
<!-- ********************************************** -->
<!-- ********************************************** -->
```

```
<!--File Name: C:\temp\highlight.xml-->
<!-- ********************************************** -->
<!-- ********************************************** -->
<HEADER>
<NAME> Moldflow Simulation Results XML Writer</NAME>
<Version> 1.00</Version>
</HEADER>
<Dataset Name="" ID="1700">
<DataType> ELDT(Highlight data)</DataType>
<DeptVar Name="Clamp force centroid" Unit=""/>
<NumberOfIndpVariables> 0</NumberOfIndpVariables>
<Blocks>
<NumberOfBlocks> 1</NumberOfBlocks>
<Block Index="1">
<Groups>
<NumberOfGroups> 28</NumberOfGroups>
<Group>
<NumberOfPoints> 2</NumberOfPoints>
<Point Index="1">
<LOC> -2.8685e-002 1.1691e-002 4.1508e-002 </LOC>
</Point>
...
...
</Group>
</Groups>
</Block>
</Blocks>
</Dataset>
```

```
</MoldflowXMLDocument>
```