Autodesk® Moldflow® Insight 2012

AMI Import and Export

Revision 1, 21 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

Supported model import formats

1

This table lists the model formats that you can import.

NOTE: The table below lists the various model formats that you can import. Note that most of these formats require additional Autodesk Moldflow Design Link software to be installed. The add-in enables you to read additional file formats.

File format	Recognized file extensions	Required software	Direct Import of native CAD format[1]
Autodesk Inventor 2012	*.ipt	Autodesk Moldflow Design Link	Yes
	*.iam	Autodesk Moldflow Design Link	Yes
CATIA V5R20 CATIA V5R20 Assembly	*.catpart, *.catproduct	Autodesk Moldflow Design Link for CATIA V5	Yes
Pro/ENGINEER Wildfire 5.0 Pro/ENGINEER Wildfire 5.0 Assembly	*.prt, *.asm	Autodesk Moldflow Design Link for Pro/ENGINEER	Yes
Parasolid V22	*.x_t,*.x_b,*.xmt_txt,*.xmb,*.xmt	Autodesk Moldflow Design Link for Parasolid	Yes
SolidWorks 2011 SolidWorks 2011 Assembly[2]	*.sldprt, *.sldasm	Autodesk Moldflow Design Link for Parasolid	Yes

[2] Requires SolidWorks to be installed.

File format	Recognized file extensions	Required software	Direct Import of native CAD format[1]
SAT[3]	*.sat	Autodesk Moldflow Design Link	Yes
STEP	*.stp,*.step	Autodesk Moldflow Design Link for Parasolid or Autodesk Moldflow Design Link for Pro/ENGINEER	No
IGES	*.igs,*.iges	None or Autodesk Moldflow Design Link for Parasolid or Autodesk Moldflow Design Link for Pro/ENGINEER	No
Autodesk study file	*.sdy	None	Not applicable
ANSYS Prep 7	*.ans	None	Not applicable
I-DEAS Universal	*.unv	None	Not applicable
NASTRAN Bulk Data	*.bdf	None	Not applicable
PATRAN Neutral	*.pat,*.out	None	Not applicable
Stereolithography	*.stl	None	Not applicable
ASCII model	*.udm	None	Not applicable

NOTE:

- When saving the model, ensure that the extension matches the entry in the table above.
- Unless the model has a *.sdy file format, only a single cavity can be imported. Multiple cavity models must be separated into single cavities before import.

[3] You can import a SAT file (versions 4.0–7.0).

Supported model import formats

You can import models of different formats into this product for analysis.

Importing a CAD model

You can import existing CAD model files to begin the part design process. Before you import a model, it is best that you first create a new project or open an existing project. When you import a CAD model, you are prompted to select the following settings:

- mesh type
- units (when an STL model is selected)

Information specific to STL is also included below.

NOTE: Autodesk Moldflow Design Link is the preferred method for importing geometry data.

1 Open an existing project or create a new project.

2 Click **Home tab > Import panel > Import**, or right-click in the **Project View** pane and select **Import**.

3 Select the correct file extension for your CAD model from the **Files of type** drop-down list.

4 Navigate to the folder where your CAD model is located and select it.

5 Click **Open**.

6 Select the appropriate mesh type from the **Import** dialog, then click **OK**.

NOTE: If you chose to import an STL model, (with extension ***.stl**) you will also need to specify the appropriate units from the **Import** dialog.

7 Click **OK**.

The model appears inside the **Project View** pane.

Importing an ASCII model file

You can import existing ASCII model files. When you import the model file, you will be prompted to select the mesh type that you want to work with.

1 Open an existing project or create a new project.

2 Click **Home tab > Import panel > Import**, or right-click in the **Project View** pane and select **Import**.

3 In the **Files of type** drop-down list, select **ASCII/Binary Model (*.udm)**.
4 Navigate to the folder where your model is located, select the file, and then click **Open**.
5 Select the appropriate mesh type from the **Import** dialog that appears.
6 Click **OK**.

Importing a model of the core from a CAD program

You can import the core model from a CAD program.

If you want the core to extend past the end of the part, use (**Inserts**) to create a mold insert.

NOTE: You can prepare the core while it is mesh if the starting point of the core is the mesh of the part, originally exported from Autodesk Moldflow Insight. This ensures the mesh of the core will match the part perfectly. If you are importing the geometry of the core created in a CAD package, it will be difficult to match the surface mesh between the core and the part.

---

1 Import the CAD model of the core into a new study, using a Dual Domain mesh with a similar density to the part model.
2 Double click the mesh icon in the **Study Tasks** pane to create the Dual Domain mesh.
3 Repair the mesh where necessary to ensure it has no errors.
4 Change the properties of the elements on the **Core elements** layer to **Part Surface (Dual Domain)**.
5 Change the mesh type to 3D and remesh the core.

TIP: Use a minimum of 4 elements through the thickness of the mesh.

NOTE: The core mesh and the part mesh must be within the Surface Matching Tolerance value (default: 0.22mm).

6 Set the **Property Type** of all elements on all layers to **Core 3D**.

TIP: The name of the property assigned will be displayed in the analysis logs. If you have multiple cores, you can apply a separate **Core 3D** property with a different name to the elements in each of the cores.

7 You can change the material of the core and the **local mold surface temperature control** by selecting all elements on all layers, then editing their properties: **Geometry tab > Properties panel > Edit**.
8 You must set a fixed constraint on the nodes at the fixed end of the core, where it joins to the mold. Click **Boundary Conditions tab > Constraints and Loads panel > Constraints > Fixed Constraint**.

9 Select all the nodes at the fixed end of the core.

TIP: Ensure that the **Select Enclosed Entities** option is set (**Geometry tab > Selection panel > Select Enclosed entities**) and that you rotate the part so the nodes you want to select are in a line. This stops unwanted nodes from being selected.

---

10 In the **Input Parameters** section of the **Fixed Constraint** tool, select **Core-shift Analysis** from the **Use constraint in** drop down box, then select **Apply** to apply the fixed constraint.

11 Rename the layers with core elements in them to:

- Core nodes
- Core tetras
- Core constraints

This prevents duplication of existing layer names, and allows you to easily identify the core layers when the core is added to the model of the part.

12 Click then click **Save > Save Study** to save your study.

13 Open the study containing the model of the part, and add the study containing the core model to it.

The model is now ready to be used in a Core-shift analysis.

Importing a Moldflow Plastics Insight 2.0 project

You can import an entire project from MPI 2.0, including the model and results files, as indicated below.

1 Right-click in the **Project View** pane and select **Import**.

The **Import** dialog appears.

2 Navigate to the location of the MPI 2.0 Projects (moldflow.prj).

3 Change the **Files of Type** to **MPI 2.0 Projects (moldflow.prj)**.

4 Click **Open**.

The **Import—Create Project** dialog appears.

5 Give the project a new name, and location if required, and click **OK**.

Supported model import formats

Select a model to import, and specify the units of measure and analysis technology for the import process.

To access this dialog, click then (**Open > Import**).

Import—Create New Project dialog

This dialog is used to create or open a project when you import a file with no project open. A project can be thought of as a container that holds all work related to a particular design.

To access this dialog, close the current project, click , and then click (**Open > Import**). Create or open a project, and then continue to locate and open the file to be imported.

Import dialog

This dialog is used to select the parameters for the model import process, in particular the target mesh type and the module to perform the import (internal or Autodesk Moldflow Design Link).

To access this dialog, click (**Import**) from the Quick Access toolbar, then locate and open the model file you want to import.

NOTE: You still have the opportunity to change the mesh type at a later time by right-clicking on the mesh type and selecting the required entry in the **Set Mesh Type** menu.

Autodesk Moldflow Design Link

Autodesk Moldflow Design Link provides a geometry data translation interface to leading CAD systems.

Autodesk Moldflow Design Link is an add-on program for Autodesk Moldflow Insight and Autodesk Moldflow Adviser, which provides a geometry data translation interface between the Autodesk Moldflow range of simulation products and leading CAD systems.

Autodesk Moldflow Design Link uses standard file formats, such as IGES and STEP, and also enables the direct import of native Inventor, Parasolid, SolidWorks, Pro/ENGINEER, and CATIA V5 part files.

After installing Autodesk Moldflow Design Link, and if you have the required licenses, a number of additional CAD file types will automatically become available on the CAD file import dialog in Autodesk Moldflow Insight or Autodesk Moldflow Adviser.

NOTE:

- The first character of the CAD file name must be a letter and not a number.
- The file name must consist of letters or numbers, and must not include dashes or any other characters.

Autodesk Moldflow Design Link

You can import a number of different CAD file types if you have Autodesk Moldflow Design Link installed.

Importing or Processing a CAD model using Autodesk Moldflow Design Link

To begin the part design process, you can import or process existing CAD model files using Autodesk Moldflow Design Link.

Before you import a model, first create a new project (or open an existing project) in which to store the model.

1. Click **Home tab > Import panel > Import**, or right-click in the **Project View** pane and select **Import**.

   The **Import** dialog appears.

2. Navigate to the folder where your CAD model is located and select it.
3. Click **Open**.
4. If you selected a native CAD geometry, do one of the following:
   a. Click **Direct Import using Autodesk Moldflow Design Link** to import the model in its native format
   b. Click **Process using Autodesk Moldflow Design Link** to translate the model.

   NOTE: If you selected this last option, click **Advanced**, select the required alternate meshing parameters that Autodesk Moldflow Design Link offers and click **OK**.

5. Select the analysis technology to be used from the drop-down list.
6. Click **OK**.

NOTE: The model appears inside the **Project View** pane.

Autodesk Moldflow Design Link

Modify the import settings used by by the Autodesk Moldflow Design Link translator.

Advanced Import Options dialog

This dialog is used to configure the model import settings used by the Autodesk Moldflow Design Link translator.

To access this dialog, click (**Import**) from the Quick Access toolbar, then locate and open the model file you want to import. Select **Process using Autodesk Moldflow Design Link** then click **Advanced**.

NOTE: If you do not have the latest version of Autodesk Moldflow Design Link installed, some features may not be available.

NOTE: This dialog is only available for file formats and file extensions supported by the Autodesk Moldflow Design Link translator.

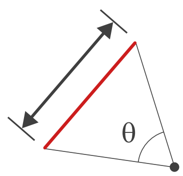
Chord angle

When importing a CAD model with Autodesk Moldflow Design Link, areas of high curvature can be selectively modeled using the **Chord angle** option.

For tightly curved sections of the model that are not meshed adequately, you can increase the mesh density in these areas by defining a chord angle.

The chord angle controls how closely the curve of the CAD model is approximated by straight sections in the mesh. The smaller the chord angle, the shorter the chord length, the finer the mesh. If the resulting chord length is greater than the general edge length, the general edge length will be used.

The chord angle can be defined by either adjusting the slider bar or entering a value into the **Tolerance** text box.



Mesh on assembly contact faces

In an assembly, faces that come into contact with each other should have aligned nodes so that the results, such as heat transfer across the surface, are as accurate as possible.

By default, a precision alignment of nodes is generated when an assembly is imported with Autodesk Moldflow Design Link. If there are errors in the imported CAD model, such as misalignment of the contact faces, faces intersecting with opposite surfaces, or boundaries not aligning, the alignment of nodes may not be possible and in some instances the meshing may fail. If this occurs, you can select **Fault tolerant** and reimport the model.

With the **Fault tolerant** option, small variations in the CAD model are compensated for, and the mesh is aligned where possible. You should check your mesh to ensure a suitable result. If the CAD model is still not suitable for this option, the model should be reimported with the **Ignore contact** option selected.

The **Ignore contact** option is used when the CAD model has errors that cannot be automaticallty resolved. In this case, a mesh will be generated for the component parts, regardless of any errors in the CAD model. The resultant mesh may have to be modified manually to improve the match on contact surfaces or remove surface intersections.

Supported IGES entities

This is a table summarizing the IGES entities supported by this product.

IGES Entity Name	IGES Entity Number	Form Number	Parasolid Entity
Circular Arc	#100	0	Circle
Composite Curve	#102	0	Curve List
Conic Arc : Ellipse	#104	1	Ellipse
Conic Arc : General	#104	0	Spline Curve
Copious Data : 2D Path	#106	11	Curve List
Copious Data : 3D Path	#106	12	Curve List
Copious Data : Closed 2D Curve	#106	63	Curve List
Plane Entity : Bounded	#108	1	Plane
Line	#110	0	Line
Point	#116	0	Point
Ruled Surface	#118	1	Spline
Surface of Revolution	#120	0	Spun surface
Tabulated Cylinder	#122	0	Swept surface
Direction	#123	0	Vector
Transformation	#124	0	Transf
Rational B-Spline Curve	#126	0	Spcurve
Rational B-Spline Surface	#128	0	Spline
Offset Curve	#130	0	Curve
Offset Surface	#140	0	Surface

IGES Entity Name	IGES Entity Number	Form Number	Parasolid Entity
Boundary Entity	#141	0	Loop
Curve on Parametric Surface	#142	0	Loop
Bounded Surface	#143	0	Face
Trimmed Surface	#144	0	Face
MSBO	#186	0	Solid
Plane Surface	#190	0	Plane
Right Circular Cylindrical Surface	#192	0	Cylinder
Right Circular Conical Surface	#194	0	Cone
Spherical Surface	#196	0	Sphere
Toroidal Surface	#198	0	Torus
Subfigure Definition Entity	#308	0	Solid
Associative Instance Entity	#402	1,7	--
Subfigure Instance Entity	#408	0	Solid
Vertex List	#502	1	Vertex
Edge List	#504	1	Edge
Loop	#508	1	Loop
Face	#510	1	Face
Shell	#514	1	Shell

Supported STEP entities

This table lists the STEP entities that are supported by Autodesk Moldflow Insight

STEP entity classes	Parasolid entity	
Topology		
MANIFOLD_SOLID_BREP	PK_BODY_t	---
CLOSED_SHELL	PK_SHELL_t	---
ADVANCED_FACE	PK_FACE_t	---
EDGE_LOOP	PK_LOOP_t	---

STEP entity classes	Parasolid entity	
Topology		
ORIENTED_EDGE	PK_FIN_t	---
EDGE_CURVE	PK_EDGE_t	---
VERTEX	PK_VERTEX_t	---
Geometry		
CARTESIAN_POINT	PK_POINT_sf_t	---
LINE	PK_LINE_sf_t	---
CIRCLE	PK_CIRCLE_sf_t	---
ELLIPSE	PK_ELLIPSE_sf_t	---
PARABOLA	PK_BCURVE_sf_t	With vertex_dim=4 in Parasolid
HYPERBOLA	PK_BCURVE_sf_t	With vertex_dim=4 in Parasolid
PLANE	PK_PLANE_sf_t	---
CYLINDRICAL_SURFACE	PK_CYL_sf_t	---
CONICAL_SURFACE	PK_CONE_sf_t	---
SPHERICAL_SURFACE	PK_SPHERE_sf_t	---
TOROIDAL_SURFACE	PK_TORUS_sf_t	---
Spline Curves		
UNIFORM_CURVE	PK_BCURVE_sf_t	With vertex_dim = 3 in Parasolid
QUASI_UNIFORM_CURVE	PK_BCURVE_sf_t	-do-
BEZIER_CURVE	PK_BCURVE_sf_t	-do-
B_SPLINE_CURVE_WITH_KNOTS	PK_BCURVE_sf_t	-do-
NURBS	PK_BCURVE_sf_t	With vertex_dim = 4 in Parasolid
TRIMMED_CURVE	CURVE_t	Base curve is mapped which gets trimmed by the boundary vertices in Parasolid
SURFACE_CURVE	CURVE_t or PK_SPCURVE_t	
INTERSECTION_CURVE	CURVE_t	Only the curve is mapped to

Spline Curves		
		corresponding curve in Parasolid
PCURVE	PK_SPCURVE_t	---

Spline surfaces		
B_SPLINE_SURFACE_WITH_KNOTS	PK_BSURF_sf_t	With vertex_dim = 3 in Parasolid
UNIFORM_SURFACE	PK_BSURF_sf_t	-do-
QUASI_UNIFORM_SURFACE	PK_BSURF_sf_t	-do-
BEZIER_SURFACE	PK_BSURF_sf_t	-do-
NURBS	PK_BSURF_sf_t	With vertex_dim = 4 in Parasolid

Others		
SURFACE_OF_LINEAR_EXTRUSION	PK_SWEPT_sf_t	---
SURFACE_OF_REVOLUTION	PK_SPUN_sf_t	---
CURVE_BOUNDED_SURFACE	PK_BODY_t	
RECTANGULAR_TRIMMED_SURFACE	PK_BODY_t	
SHELL_BASED_SURFACE_MODEL	PK_BODY_t	
FACETED_BREP	PK_BODY_t	
OFFSET_SURFACE	PK_OFFSET_sf_t	
BLEND	PK_BSURF_sf_t	
SPUN_SURFACE	PK_SPUN_sf_t	

Using models imported from Autodesk Simulation products

Autodesk Simulation products are used to perform a structural analysis on a part.

Incorporating the material properties and flow-induced characteristics of a plastic part enhances the structural analysis results.

When a part is exported from Autodesk Simulation to Autodesk Moldflow Insight, the polymer defined while the model is being prepared in the Autodesk Simulation product is imported and by default is selected for the Autodesk Moldflow Insight analysis.

As with all models imported into Autodesk Moldflow Insight, the Autodesk Simulation model still needs to be prepared for an analysis. At minimum, you must set the mesh type and generate the mesh, and you must set at least one injection location on the model.

NOTE: This feature is available for **Thermoplastics Injection Molding** analyses of Dual Domain and 3D models.

NOTE: In order for Autodesk Moldflow Insight results to transfer successfully for the structural analysis, you cannot change the orientation of the imported model with respect to the global coordinate system or modify the geometry. You should not change the name of the study that was created on import.

Using models imported from Autodesk Simulation products

Models imported from Autodesk Simulation structural analysis products need to be prepared for an analysis.

Using Autodesk Simulation products with Autodesk Moldflow Insight

Models imported from Autodesk Simulation products only have a surface geometry representation and so need to be meshed.

NOTE: This feature is available for **Thermoplastics Injection Molding** analyses of Dual Domain and 3D models.

1. Click **Mesh tab > Mesh panel** and select **Dual Domain** or **3D**.
2. Click **Mesh tab > Mesh panel > Generate Mesh** to mesh the model for Autodesk Moldflow analysis. If necessary, you can change global or local mesh density and remesh the model.
3. Click **Home tab > Molding Process Setup panel > Injection Location** and define one or more injection locations.

   NOTE: For analyses of parts where structural performance criteria are critical, it is important that the injection location used in the analysis should match the injection location used in production.

4. Click **Home tab > Molding Process Setup panel > Analysis Sequence** and select an analysis sequence that includes **Fill + Pack**.
5. Click **Home tab > Analysis panel > Start Analysis** to launch the analysis.

When the analysis is complete, the resulting process-induced mechanical properties automatically are made available to Autodesk Simulation for use in the structural analysis.

NOTE: In order for Autodesk Moldflow Insight results to transfer successfully for the structural analysis, you cannot change the orientation of the imported model with respect to the global coordinate system or modify the geometry. You should not change the name of the study that was created on import.

Importing IGES model files

This is a table summarizing the IGES entities supported by this product.

IGES files have the extension *.igs or *.iges. This table lists the IGES entities supported by the file import feature in Autodesk Moldflow Insight. The supported IGES versions are 5.3 or earlier.

After importing an (*.igs) file, and before running a Fill+Pack analysis, pay close attention to the edge length of the mesh around high curvature areas on your model, and make sure they are not too coarse. It is recommended you mesh with a smaller edge length allowing the mesh to approximate the corners correctly.

NOTE: Autodesk also supplies Autodesk Moldflow Design Link which has extended IGES import capabilities and the ability to import native file formats.

Supported IGES entities

The following table lists the IGES entities that are recognized and translated when reading in an IGES file. If the IGES file contains entities that are not listed here, they will be ignored in the translation process.

IGES entity name	IGES entity number	Form number	Parasolid entity
Circular Arc	100	0	Circle
Composite Curve	102	0	Curve List
Conic Arc: Ellipse	104	1	Ellipse
Conic Arc: General	104	0	Spline Curve
Copious Data: 2D Path	106	11	Curve List
Copious Data: 3D Path	106	12	Curve List
Copious Data: Closed 2D Curve	106	63	Curve List
Plane: Bounded	108	1	Plane
Line	110	0	Line

IGES entity name	IGES entity number	Form number	Parasolid entity
Parametric Spline Curve	112	0	
Parametric Spline Surface	114	0	
Ruled Surface	118	1	Spline
Surface of Revolution	120	0	Spun Surface
Tabulated Cylinder	122	0	Swept Surface
Direction	123	0	Vector
Transformation Matrix	124	0	Transf
Rational B-Spline Curve	126	0	Spcurve
Rational B-Spline Surface	128	0	Spline
Offset Curve	130	0	Curve
Offset Surface	140	0	Surface
Boundary Entity	141	0	Loop
Curve on a Parametric Surface	142	0	Loop
Bounded Surface	143	0	Face
Trimmed Surface	144	0	Face
MSBO*	186	0	Solid
Plane Surface*	190	0	Plane
Right Circular Cylindrical Surface*	192	0	Cylinder
Right Circular Conical Surface*	194	0	Cone
Spherical Surface*	196	0	Sphere
Toroidal Surface*	198	0	Torus
Subfigure Definition Entity	308	0	Solid

IGES entity name	IGES entity number	Form number	Parasolid entity
Associative Instance Entity	402	1,7	--
Singular Instance Entity	408	0	Solid
Vertex List*	502	1	Vertex
Edge List*	504	1	Edge
Loop*	508	1	Loop
Face*	510	1	Face
Shell*	514	1	Shell

* Requires Autodesk Moldflow Design Link to be installed.

Importing IGES model files

To obtain an accurate analysis, it is important that the IGES model is imported correctly.

Guidelines for preparing IGES files for import

To successfully translate an IGES model into an Autodesk model suitable for analysis, the model must have been correctly prepared in the CAD system.

- The entire model must be described by IGES surfaces, not just lines and curves.
- Lines and curves can be used to import the cooling channels of a mold, and can be used as the basis for cooling channels construction. You may be able to export the center line geometry of the cooling lines from the CAD package.
- If possible, simplify the model to remove unnecessary detail, such as reference planes and very small features that have no effect on a Fill+Pack or a Stress analysis.
- Before exporting a model to be used by Dual Domain analysis technology, check in the CAD system that the part is fully closed, i.e. no gaps between surfaces.
- Export as surfaces, not shells.

Flow leader and deflector surfaces

Flow balance, where the extremities of the product fill at the same time, is achieved by changing the thickness of flow leader or deflector surfaces. The location and shape of such surfaces can be estimated and introduced before running a Fill+Pack analysis.

Importing STL model files

This topic lists the requirements for importing stereolithography (*.stl) files.

1 The STL file should be complete and incorrupt.

An ASCII .stl file must start with the lower case keyword solid and end with endsolid.

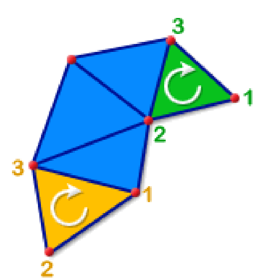
For example:

```
solid
...
facet normal 0.00 0.00 1.00
outer loop
vertex 2.00 2.00 0.00
vertex -1.00 1.00 0.00
vertex 0.00 -1.00 0.00
endloop
endfacet
...
endsolid
```

2 The STL file can be in either ASCII or binary format. It is important to use the correct format with FTP. For example, if you have a binary STL file, you must set the FTP file type to “binary” before transferring.

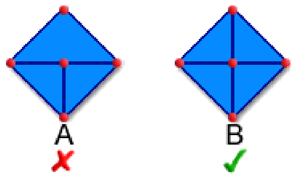
3 Only one solid should be present.

4 The triangles should be defined clockwise, with the normal indicating the "out" direction:

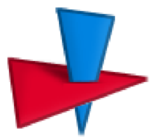


5 The orientation of the triangle normals should be aligned.

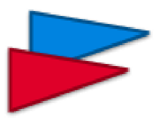
6 The triangles should share common corner node positions (the “vertex to vertex” rule). There should be no gaps or free edges in the mesh of triangles.



7 There should be no intersections between the triangles' surfaces and, naturally, edges should overlap).



8 There should be no triangle overlaps.



9 As a guide (not a requirement), there should be no more than 20,000 facets in one model.

If you can, use the STL settings in your CAD system to base the resolution of the *tessellations* on the chord height calculation below. You can thus reduce the number of triangles.

$C=M1000\times Q$

where:

- **C** = chord height
- **p** = part surface
- **t** = tessellated surface
- **M** = model size (the distance between opposite diagonals of the bounding box of the part)
- **Q** = quantity of elements (recommended 0.3, limits 0.1 to 1.0)

NOTE: Q is determined by the user and describes the size of the facets, especially in areas of high curvature. A higher Q value will result in a larger number of smaller facets. A large number of facets may take longer to analyze.

10 If your STL model has triangles with very high aspect ratio, the mesh will be distorted, and the analysis results will be less accurate.

For a mesh triangle, the aspect ratio is the ratio of the length of the longest side (a) to the height perpendicular to that side (b). As a general rule, this ratio should be less than 6:1.

The program can accept some triangles with very high aspect ratios (hundreds or even thousands). However, try to keep the average aspect ratio below 6.

Importing STL model files

How to import STL file types.

Importing an STL Midplane

An STL Midplane mesh is not a standard format. However, if you need to import a Midplane mesh that is stored in STL format, follow these steps:

1. Import the STL. In the **Import** dialog, set the model type to Dual Domain, not Midplane, and then click **OK**.
2. Click **Mesh tab > Mesh panel > Generate Mesh** and create a mesh on your model.

   This process will remesh the STL surface into a Midplane mesh (but the type will still be set to Dual Domain).
3. When the **Mesh complete!** dialog appears, click **OK**.
4. In the **Study Tasks** pane, right-click **Dual Domain Mesh** to change the mesh type to **Midplane**.
5. Assign properties (such as part surface) to the newly-created mesh.

Importing an ASCII model file

You can import existing ASCII model files. When you import the model file, you will be prompted to select the mesh type that you want to work with.

1. Open an existing project or create a new project.
2. Click **Home tab > Import panel > Import**, or right-click in the **Project View** pane and select **Import**.
3. In the **Files of type** drop-down list, select **ASCII/Binary Model (*.udm)**.
4. Navigate to the folder where your model is located, select the file, and then click **Open**.

5 Select the appropriate mesh type from the **Import** dialog that appears.
6 Click **OK**.

Importing ANSYS model files

ANSYS model files have the extension *.ans. This help topic lists the entity and element types supported by Autodesk Moldflow Insight.

Supported entities

The supported entities are:

Entity Type	Description
NBLOCK	block formatted nodes
EBLOCK	block formatted elements
EN,R5.0	element card
EN,R5.1	element card
EN,R5.5	element card
EN,4.4	element card
E,	element card
EN,	element card
R,R5	real constant tables
R,	real constant table

The supported element types are:

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

Importing IDEAS universal model files

This help topic provides a detailed list of the SDRC/I-DEAS entities that are supported by Autodesk Moldflow Insight. SDRC/I-DEAS universal files have the extension *.unv and can be used in a 3D analysis.

Supported Universal Datasets

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

Dataset	Description
151	Header
164	Units
776	Beam Cross Sections (circular only)
2411	Nodes
2412	Elements
2429	Permanent Groups
2437	Physical Properties
2448	Physical Properties

NOTE: For further details about the boundary condition related datasets supported by the I-DEAS / Autodesk integration, please refer to your I-DEAS documentation.

Obsoleted Datasets

The following tables lists those datasets supported in previous releases that are now no longer supported.

Obsolete Dataset	Description
15	Nodes
772	Physical Props
780	Elements
781	Nodes
789	Physical Props
Boundary condition datasets... (1)	
792	Temperature Sets
833	Mold Filling
835	Mold Cooling
836	Shell Thickness Data
840	Thermoplastic Data
843	Gate Node, runnerless model
1714	Materials Database Material
1750	Materials Database Material

Importing NASTRAN bulk data model files

NASTRAN Bulk Data files have the extension (*.bdf). This help topic lists the NASTRAN element types that are supported by Autodesk Moldflow Insight.

NASTRAN Element Types

The following NASTRAN element types can be read (others will be ignored):

Element type
GRID
CTRIA
CTRIA3
CQUAD
CQUAD4
CQUAD8
CQUADR
CQUADX
CTETRA
CBAR (Beam elements)
PSHELL

Restrictions

Only models described in absolute coordinates can be read in.

Importing PATRAN neutral model files

PATRAN Neutral files supported by Autodesk Moldflow Insight have the extension *.pat, or *.out. This help topic lists the supported PATRAN element types and how to pass thickness values to Autodesk Moldflow Insight.

Supported PATRAN Element Types

The following PATRAN entities or element types can be read:

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

Defining Element Property Data

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

Field	Property Data
1	not used
2	thickness
3	not used

NOTE: Element property data written by CAD systems will vary between systems. Contact your CAD supplier if you are not sure what fields contain what element property data (attributes).

Restrictions

Only models described in absolute coordinates can be read into Autodesk Moldflow Insight.

NOTE: The **Analysis Preference** export option in Patran affects the output neutral file (*.out). Sometimes, if you use the **MARC** preference and import the neutral file, there will be no thicknesses assigned. Using the **Abaqus** preference to define the shell thickness results in correct importation of thickness.

Importing a C-MOLD *.fem file

Import C-MOLD *.fem files (pre-dating C-MOLD 2000) to use them in an analysis.

Importing a C-MOLD *.fem file

If you have a C-MOLD *.fem file (pre-dating C-MOLD 2000) and want to use it in an analysis, follow the steps below to import and convert the file so it is suitable.

Importing a C-MOLD *.fem file for Microchip Encapsulation

If you have a C-MOLD *.fem file (pre-dating C-MOLD 2000) and want to use it for a Microchip Encapsulation analysis, follow the steps below to import and convert the file so it is suitable for analysis.

NOTE: Not supported for 3D models.

1 Click (**Import**) from the Quick Access toolbar, and import a CAD model.

   Alternatively, click then **Open > Import** to import the C-MOLD *.fem mesh file.

2 Identify the wire elements:

   a In the **Layers** pane, click **New Layer** and name the new layer **Wire**.

   b Click **Select**.

   c On the model, select the wire elements.

   d In the **Layers** pane, click **Assign Layer**.
   The wire elements will be assigned to the **Wire** layer.

   e Ensure the wire elements are still selected.

   f Click **Mesh tab > Properties panel > Change**.

   Alternatively, right-click and select**Change Property Type**

   g In the **Change Property To** dialog, select **Wire**.

3 Identify the leadframe elements:

   a In the **Layers** pane, click **New Layer** and name the new layer **Leadframe**.

   b Click **Select**.

   c On the model, select the leadframe elements.

   d In the **Layers** pane, click **Assign Layer**.
   The leadframe elements will be assigned to the **Leadframe** layer.

   e Ensure the leadframe elements are still selected.

   f Click **Mesh tab > Properties panel > Change**.

   Alternatively, right-click and select**Change Property Type**

   g In the **Change Property Type To** dialog, select **Leadframe**.

4 Click then **Save > Save Study** to save the changes.

Importing a C-MOLD *.fem file

If you have a C-MOLD *.fem file (pre-dating C-MOLD 2000) and want to use it for an Underfill Encapsulation analysis, follow the steps below to import and convert the file so it is suitable for analysis.

NOTE: Not supported for 3D models.

---

1 Click (**Import**) from the Quick Access toolbar to import the C-MOLD *.fem mesh file.

2 Identify the wire elements:

   a In the **Layers** pane, click **New Layer** and name the new layer **Wire**.
   b Click **Select**.
   c On the model, select the wire elements.
   d In the **Layers** pane, click **Assign Layer**.
      The wire elements will be assigned to the **Wire** layer.
   e Ensure the wire elements are still selected.
   f Click **Mesh tab > Properties panel > Change**.
      Alternatively, right-click and select**Change Property Type**
   g In the **Change Property Type To** dialog, select **Wire**.

3 Identify the leadframe elements:

   a In the **Layers** pane, click **New Layer** and name the new layer **Leadframe**.
   b Click **Select**.
   c On the model, select the leadframe elements.
   d In the **Layers** pane, click **Assign Layer**.
      The leadframe elements will be assigned to the **Leadframe** layer.
   e Ensure the leadframe elements are still selected.
   f Click **Mesh tab > Properties panel > Change**.
      Alternatively, right-click and select**Change Property Type**
   g In the **Change Property Type To** dialog, select **Leadframe**.

4 Click then **Save > Save Study** to save the changes.

Correcting baffles imported from C-MOLD 2000

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

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

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

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

1 In the **Layers** pane, turn off all layers other than the one to which the baffle was assigned when it was imported.
2 Select the baffle and then click **Geometry tab > Properties panel > Edit**.

   Ensure that the baffle has been assigned the property **Baffle** and that it has a Heat Transfer Effectiveness=**0.5**.
3 Click **Geometry tab > Utilities panel > Move > Translate**.
4 In the model pane, click on the element representing the baffle.
5 Select **Copy**, specify the vector **0 , 0 , 0** and click **Apply**.

   This will create a copy of the element, superimposed on top of the existing one.
6 Click **Mesh tab > Mesh panel > Density**. Set the mesh density to 2.5 times the diameter of your baffle. Click **Apply** and **OK**.
7 Click **Mesh tab > Mesh panel > Generate Mesh**, deselect the check boxes and then click **Mesh Now**.

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

Correcting bubblers imported from C-MOLD 2000

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

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

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

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

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

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

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

    The inner diameter of the bubbler must be equal to or greater than that of the inner channel.
11. Click **Geometry tab > Utilities panel > Move > Translate**.
12. In the graphics pane, click on the new bubbler you have made.
13. In the **Move/Copy—Translate** dialog, reverse the sign of each of the vector components you specified earlier and click **Apply**.

    This will move the element so that the bubbler is superimposed on the channel element.
14. Click **Mesh tab > Mesh panel > Density**. Set the mesh density to 2.5 times the diameter of your bubbler. Click **Apply** and **OK**.
15. Click **Mesh tab > Mesh panel > Generate Mesh**, deselect the check boxes, and click **Mesh Now**.

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

Exporting models and files

2

You can export data in different formats and import the files into other software.

The supported export formats and their functions are summarized in the following table.

Output format (extension)	Entities exported	Purpose for export	Available for...
Autodesk Moldflow Results file (*.mfr)	Selected study results[4]	Share results using Autodesk Moldflow Communicator	Midplane Dual Domain 3D
Zip Archive (*.zip)	Entire project or selected studies	Archive studies, or zip project for colleagues or Technical Support	Midplane Dual Domain 3D
Altair Hyper3D (*.h3d)	Meshed model and study results	Provide mesh and result data directly in the format supported by Altair Engineering's HyperView 9.0 visualization products[5]	Midplane Dual Domain 3D
ASCII FBX File (*.fbx)	Selected model entities and study results	Export part surface and result data to Autodesk Showcase software for photo-realistic visualization	Dual Domain 3D
ASCII Model File (*.udm)	Entire study contents	Export the study to an ASCII format file for	Midplane Dual Domain

[4] Before an Autodesk Moldflow Results file (*.mfr) can be generated, the selected results must be marked for export in the project.

[5] HyperView 9.0 displays mesh and result data associated with mesh nodes or elements. HyperView Player 9.0 displays mesh only. Earlier versions of HyperView and HyperView Player cannot display exported *.h3d files.

Output format (extension)	Entities exported	Purpose for export	Available for...
		support diagnosis or for data transfer	3D
Patran File (*.pat)	Nodes, triangles, beams, tetrahedra	Import the model into a 3rd party CAE system	Midplane Dual Domain 3D
M3I File (*.m3i)	3D study files	MPI 2.0 format for storing 3D mesh	3D
MFL File (*.mfl)	Nodes, triangles, beams, basic properties	Obsolete format—provided for compatibility with MPI 2.0	Midplane Dual Domain 3D
ASCII IGES File (*.igs)	NURBS surfaces, permitted NURBS curves	Import the model in a 3rd party CAD system for editing purposes	Midplane Dual Domain 3D
Surface mesh for AMA (*.amm)	Triangles[6]	Prepare a surface mesh to be imported into Autodesk Moldflow Adviser	Dual Domain
SAT v7 (*.sat)	CAD model[7]	Export a CAD model to a SAT v7 format file to allow geometry modification using Autodesk Inventor Fusion	Dual Domain 3D

Table 1: Permitted NURBS curves

IGES entity no.	Description
102	Composite Curve
126	Rational B-Spline Curve
128	Rational B-Spline Surface

[6] A Surface mesh for AMA (*.amm) file only includes the portion of the Dual Domain model that is meshed with triangles; typically this is the part only. Other model entities, such as beams, and boundary conditions, such as injection locations, are not included in this export file format.

[7] The study must contain at least one imported CAD model in a supported native geometry format in order to export the geometry to the SAT v7 format.

IGES entity no.	Description
142	Curve on a Parametric Surface
144	Trimmed Surface

Exporting models and files

You can export data in different formats and import the files to other software.

Exporting files

1. Select the appropriate study (*.sdy) file(s), if the whole project is not to be exported.
2. Click **Results tab > Export and Publish panel > Moldflow Results**.

   Alternatively, click then **Export > Study & Results** .

   The **Export** dialog opens.
3. Navigate to the location where you want the export file to be saved.
4. Select the export format in the **Save as type** drop-down according to the table above.
5. In the **File name** text box, specify a name for the export file.
6. Click **Save**.
7. If you have selected to export to a Zip Archive, select the desired options in the **Export Project to ZIP Archive** dialog and click **OK**.

   The archiving will take a moment to process.

Exporting the project to a ZIP file

NOTE: Active analyses are displayed in the **Job Manager** as **[Running][x%]**. It is important these analyses are complete before proceeding.

1. Enter a File name for the Zip file, change the **Save as type** to **Zip Archive (*.zip)**. You may also want to take note of the location where the Zip archive is saved for later retrieval.
2. Click **Save**.

   The **Export Project to Zip** dialog opens.
3. Select the most relevant options for your output, and then click **OK**.

   A confirmation dialog appears and informs you the archive was successful.
4. Click **OK**.

Exporting an ASCII model file

You can export model files as ASCII format with the extension (*.udm). When exporting a model as an ASCII file, keywords, materials, and unused property sets will not be exported. Everything else will be exported to the specified file. Choose to export a *.udm file to make data available for Moldflow Manufacturing Solutions products.

1 Click **Results tab > Export and Publish panel > Moldflow Results**.

  Alternatively, click then **Export > Study & Results** .

  The **Export** dialog opens.

2 Navigate to the location where you want to store the model.

3 In the **Save as type** drop-down list, select **ASCII Model File (*.udm)**.

4 In the **File name** box, enter a name of the model, and then click **Save**.

5 Click **OK**.

Exporting results to an Autodesk Moldflow Results file

You can create a results file to view in Autodesk Moldflow Communicator when you export the results in Autodesk Moldflow Results file (*.mfr) format.

An Autodesk Moldflow Criteria file (*.criteria) may be included in a results file. You can use this to limit the information displayed when comparing results to criteria in Autodesk Moldflow Communicator.

Analysis logs are marked for export by default. If you do not want to include analysis logs in the Autodesk Moldflow Results file, right-click on **Logs** in the **Study Tasks** pane, and then select **Unmark for export**.

1 Right-click a result name in the list of results and select **Mark for export**.

  An asterisk appears after the result name. It has now been marked for export.

2 Repeat to mark additional results for export in the same way.

TIP: You can export the same results from 2 studies by marking the results in each of the studies, and then using the Control or Shift key to select both studies. Continue with the steps below to create the Autodesk Moldflow Communicator results file with results from both studies.

3 Click **Results tab > Export and Publish panel > Moldflow Results**.

4 Navigate to the location in which you want to save the results file.

5 In the **File name** text box, enter a name for the results file.

6 Select **Autodesk Moldflow Results file (*.mfr)** in the **Save as type** drop-down list, and then click **Save**.

   A confirmation dialog appears to inform you the results were exported successfully.

7 Select the **Include criteria file** check box to include a criteria file. Select the **Browse** button to choose your criteria file, and then click **Save**.

   Criteria should be specifically entered for the study and results that you are exporting.

   The file you selected is displayed in the check box.

8 If you want to restrict the display of information in Autodesk Moldflow Communicator when comparing studies to criteria, select the **Restrict MFR contents based on criteria** check box.

   When a study is compared to a restrictive criteria file in Autodesk Moldflow Communicator, only the information that is specified in the criteria file is displayed.

9 Click **OK** to export the Autodesk Moldflow Results file.

   A confirmation dialog appears to inform you the results were exported successfully.

NOTE: You can deselect all results that have been marked for export by right-clicking on any result, and then selecting **Unmark All for Export**.

Exporting a surface mesh for use with Autodesk Moldflow Adviser

You can export a **Surface mesh for AMA (*.amm)** file to prepare a Dual Domain model for import into Autodesk Moldflow Adviser software.

The current study must include a Dual Domain meshed model for the **Surface mesh for AMA (*.amm)** export format option to be available.

This format allows you to export a known surface mesh from Autodesk Moldflow Insight for import into Autodesk Moldflow Adviser.

1 Click then **Export > Model** .
2 Navigate to the location where you want to store the exported file.
3 In the **Save as type** drop-down list, select **Surface mesh for AMA (*.amm)**.
4 In the **File name** box, enter a name for the exported file, and then click **Save**.
5 Click **OK**.

The new *.amm file is available for import into Autodesk Moldflow Adviser and subsequent analysis without any changes to the mesh.

NOTE: A **Surface mesh for AMA (*.amm)** file only includes the portion of the Dual Domain model that is meshed with triangles; typically this is the part only. Other model entities, such as beams, and boundary conditions, such as injection locations, are not included in this export file format.

Exporting CAD geometry to a SAT v7 file for use with Autodesk Inventor Fusion

To export CAD solid geometry to a SAT v7 (*.sat) file for use with Autodesk Inventor Fusion, the study must contain one or more imported CAD models in a supported native geometry format. CAD geometry formats that can be exported to a SAT v7 file include:

- Autodesk Inventor 2012, IPT
- Autodesk Inventor 2012, IAM
- SAT v4–v7
- CATIA® V5R20
- Parasolid® V22
- Pro/ENGINEER® Wildfire® 5.0
- SolidWorks® 2011

1. Open a study that contains at least one imported CAD model.
2. Click (**Application menu > Export > Model**).
3. Navigate to the location where you want the export file to be saved.
4. From the **Save as type** drop-down list, select **SAT (*.sat)** file format.
5. In the **File name** text box, specify a name for the export file.
6. Click **Save**.

   If more than one CAD model exists in the study, a numerical suffix will be added to the specified file name, and each CAD model will be saved to a separate file.

   NOTE: Assemblies contain more than one CAD body but are treated as a single model. If you select a CAD body that is a component of an assembly, the entire assembly will be exported to a single SAT file.

   A message is displayed to indicate that the exported file has been saved successfully.

   NOTE: If no supported CAD model is found in the study, an error message is displayed.

You can open the exported file in Autodesk Inventor Fusion to modify the geometry.

Exporting models and files

Selected items or the entire project can be exported to various file formats.

To access this dialog, click , then click **Export** and select the desired export format.

MFR Export Settings dialog

This dialog is used to include an optional Autodesk Moldflow Criteria file (*.criteria) when exporting an Autodesk Moldflow Results file (*.mfr). To access this dialog, click (**> Export > Study and Results**), enter a name for your results file, then click **Save**.

Criteria files can be used to limit the information displayed in Autodesk Moldflow Communicator when you compare a study to criteria, using the criteria file included in the results file.

NOTE: You can also open this dialog by clicking the **Set MFR Options...** button from the **External applications tab** of the **Options** dialog (**> Options > External applications tab > Set MFR Options...**). This can be used to set a criteria file to be included by default whenever you export a results file.

STL Export Units dialog

This dialog is used to specify the units of the STL file to which you are exporting the Dual Domain mesh.

To access this dialog, ensure that the current study has been meshed with triangular elements, click , and then click (**Export > Model**), select **ASCII STL File** in the **Save as type** drop-down, navigate to the folder where you want to create the STL file, specify the name of the file to be created and click **Save**.

Export Project to ZIP Archive dialog

Export all or selected project components to a ZIP format archive file.

This feature is useful for archiving projects or for packaging a project into a single file, for example to send to a colleague or Technical Support.

NOTE: Sharing information with colleagues can also be achieved using Autodesk Moldflow Communicator a free results viewer available from the Autodesk website (*www.autodesk.com*).

To access this dialog, click , then click **Export** and complete the **Export** dialog.

TIP: Compact your projects first to remove restart files and produce a smaller archive.

Autodesk Moldflow Insight to Abaqus interface (Midplane)

Fill+Pack and Fiber results for Midplane models can be exported to a partial Abaqus input file for further structural (usually Stress) analysis.

The finite element model and result data provided by the Autodesk Moldflow Insight analysis allows Abaqus/Standard to perform shrinkage and warpage analyses based on residual stresses from the molding analysis, including filling and packing and/or fiber orientation. In addition, Abaqus/Standard can perform structural analyses on components with or without residual stress.

Interfacing to Abaqus

The **Abaqus options** dialog in the solver parameters provides three export options:

- For original Autodesk Moldflow Insight / Abaqus 6.2 license holders, a single *.mab file in ASCII format is created.
- For original C-MOLD / Abaqus 6.2 license holders, two ASCII files are generated: a *.fem finite element model file, and an *.osp file containing residual stress and/or material property information.
- For Autodesk Moldflow Insight / Abaqus 6.3 users, an *.osp file containing residual stress and/or material property information is generated. You must export the mesh model in *.pat file form (Click **File > Export** and select **Patran File (*.pat)** in the Save as Type list).

NOTE: If the *.osp file generated by Autodesk Moldflow Insight contains the "~" character; rename the file to remove that character otherwise, Abaqus will not be able to import the file.

The Abaqus Interface for Autodesk Moldflow reads the interface file and creates the following files:

Abaqus input (.inp) file	Contains model data such as nodal coordinates, element topology, and section definitions. If you are working with isotropic materials, the input file also contains material property data.
Neutral (.shf) file	Contains lamina material data for each layer of each element. (This file is created only when working with layered, spatially varying materials.)
Initial stress (.str) file	Contains initial stress data. (When executing Abaqus/Autodesk Moldflow, the user can request that this file be omitted and initial stress data ignored.) If applicable, the neutral file and the initial stress file are read into Abaqus/Standard during the initial step.

Assumptions

When working with layered, spatially varying materials, the Abaqus Interface for Autodesk Moldflow assumes the following:

- Laminated composite shell elements.
- Lamina material properties are given in the principal material direction of each layer.
- 20 laminates will be output.

NOTE: Fill+Pack outputs 12 laminates by default and will therefore be interpolated to 20 laminates for Abaqus.

When working with isotropic materials, the Abaqus Interface for Autodesk Moldflow assumes the following:

- Homogeneous shell elements.
- In the .mab file, thermo-mechanical properties of the first element are used for all elements of the model. In the .osp file, a single set of thermo-mechanical properties is provided.

Autodesk Moldflow Insight to Abaqus interface (Midplane)

Fill+Pack and Fiber results for Midplane models can be exported to a partial Abaqus input file for further structural (usually Stress) analysis.

Exporting Midplane data to Abaqus from the user interface

NOTE: This feature is only available for Midplane models on PC.

1. Ensure that you are using a Midplane model.
2. Ensure that you have selected an analysis sequence that includes Fill+Pack.
3. Click **Home tab > Molding Process Setup panel > Process Settings**.
4. If necessary, click **Next** one or more times to navigate to the **Fill+Pack Settings** page.
5. Click **Advanced Options**.
   The **Fill+Pack Analysis Advanced Options** dialog appears.
6. Click **Edit** next to the Solver Parameters field.
   The **Thermoplastics injection molding solver parameters (Midplane)** dialog appears.
7. Select the **Interface** tab and click **Abaqus options**.
   The **Abaqus Options** dialog appears.
8. Select the appropriate interface combination from the drop-down list.
9. Click **OK**.

Autodesk Moldflow Insight to Abaqus interface (3D)

Fill+Pack (with or without Fiber) and Warp results for 3D models can be exported to Abaqus for further structural (usually Stress) analysis.

NOTE: This feature is only available for 3D models on PC.

An API script (PC only) which automatically converts the necessary result and mesh files into a format that Abaqus can use is available.

The Autodesk Moldflow Insight results are exported in *.xml format and the 3D mesh is exported in Abaqus input (*.inp) format. The API script saves the interface files in a folder created in the current project folder. For example, in Windows XP, **My Documents\My AMI xxxx Projects\Project1\<study_name>_interface_files**, or in Windows Vista, **Documents\My AMI xxxx Projects\Project1\<study_name>_interface_files**,where xxxx is the version number of the Autodesk Moldflow Insight software you are using and *<study_name>* is the name of the study you are exporting to Abaqus).

Interfacing to Abaqus

Autodesk Moldflow Insight result and mesh files are binary files. In order to be used in Abaqus, these files need to be converted to ASCII format. Autodesk Moldflow Insight uses an API script, **mpi2abq.vbs**, to convert these files automatically.

Interface files

There are several types of file that Autodesk Moldflow Insight can export for use in Abaqus. The actual files exported depends on what your Autodesk Moldflow Insight study file contains.

Mesh file

The API script converts the 3D mesh to the Abaqus input format. This interface file is named **<study_name>_mesh.inp**. For more information about the Abaqus *.inp format, please refer to your Abaqus manual.

NOTE: Change from Patran to Abaqus input file for the mesh model:

Before the release of MPI 6.0, the Patran (*.pat) file format was used to convert Autodesk Moldflow Insight mesh models for input to Abaqus. Some inconsistencies were found between the mesh model in *.pat format and the result data (*.xml) files produced by the original 3D Abaqus Interface. For example:

- If there was a short shot predicted, results would not exist on the unfilled elements of the mesh model. Abaqus requires consistency between the mesh model and the corresponding *.xml files.
- If the mesh model contained mold exterior surface mesh, cooling channels, and/or a runner system, Fill+Pack analysis results would correspond only to elements on the part cavity. Abaqus would fail this model because of the inconsistency between the mesh model and result files.

Further, to interface 3D Overmolding, Microchip Encapsulation, or to interface 3D Warp results using mesh aggregation to Abaqus, the *.pat file format is of limited utility.

Beginning in MPI 6.0, the 3D Abaqus Interface converts the 3D mesh to the Abaqus input format. In this way, Autodesk Moldflow Insight can output the three point constraint for removal of the rigid body movement, build contact surface conditions between multiple components, and pass pressure and temperature conditions directly into the Abaqus input file.

Material properties data files

For fiber-filled materials, these files are produced:

Principal directions

The principal fiber orientation directions are the eigenvectors of the fiber orientation tensor, and the eigenvalues of the fiber orientation tensor representing the probability percentage of fibers aligning in the principal corresponding directions. These data are stored in **<study_name>_principalDirections.xml**.

NOTE: Change from fiber orientation tensor to principal directions:

Before the release of MPI 6.0, the fiber orientation tensor passed to the 3D Abaqus Interface, and this data together with the mechanical and thermal expansion coefficient distributions was output by the interface as *.xml files. This caused some inconvenience in data conversion, so beginning in MPI 6.0, Autodesk Moldflow Insight directly outputs the principal directions of the material property set.

Mechanical properties
Are element-based results stored in individual engineering constant component files, such as **<study_name>_E11.xml**, **<study_name>_E22.xml**, **<study_name>_E33.xml**, **<study_name>_v12.xml**, ..., and these are nine components in principal directions based on the orthotropic assumption. The calculations of these mechanical properties are based on the selected micro-mechanics model and a 9-constant fiber orientation average method along with a selected closure approximation option, which are specified in the **Fiber parameters** of the Fill+Pack process settings.

Thermal expansion coefficients
Are element-based results stored in **<study_name>_ltec_1.xml**, **<study_name>_ltec_2.xml** and **<study_name>_ltec_3.xml**, representing the linear thermal expansion in the first, second, and third principal directions. These values are calculated based on the selected method in the **Fiber parameters** of the Fill+Pack process settings, with an orientation average.

For unfilled materials, these files are produced:

- **<study_name>_Moduli.xml**
- **<study_name>_PoissonRatios.xml**
- **<study_name>_ShearModuli.xml**
- **<study_name>_Ltecs.xml**

Initial stresses file

Autodesk Moldflow Insight passes the initial stresses calculated by the 3D Warp analysis to the interface. The API script converts this data and stores it in **<study_name>_initStresses.xml**.

NOTE: Change from volumetric shrinkage to initial stress data:

Before the release of MPI 6.0, the 3D Abaqus Interface translated the volumetric shrinkage result from 3D Flow, using the **strintf3d** script, to a format that could be converted into initial stresses by Abaqus' Autodesk Translator (through a command ***INITIAL CONDITIONS, TYPE=STRESS, USER**). However, because 3D Warp calculates the initial stresses internally using a proprietary technology, the inconsistency in the final warp results predicted by 3D Warp and by Abaqus could easily be identified. For this reason, beginning in the MPI 6.0 release, the initial stress data calculated by 3D Warp is passed directly to the 3D Abaqus Interface.

Unit conversion

Before the release of MPI 6.0, the interface files were always produced in the SI unit system, and a special script had to be coded to take care of unit conversion. Beginning in the MPI 6.0 release, the **mpi2abq** script takes care of the unit conversion if you select a unit system other than SI.

Autodesk Moldflow Insight to Abaqus interface (3D)

Fill+Pack (with or without Fiber) and Warp results for 3D models can be exported to Abaqus for further structural (usually Stress) analysis.

Exporting 3D data to Abaqus from the user interface

NOTE: This feature is only available for 3D models on PC.

There are two parts to exporting Fill+Pack, Fiber and Warp results for 3D models to Abaqus: the first is running the **mpi2abq.vbs** macro to create the necessary files; the second is locating the files and using them in Abaqus.

To run the **mpi2abq.vbs** macro from the user interface:

1. Click **Tools tab > Automation panel > Play Macro**.
2. In the **Open Macro** dialog, navigate to the location where the **mpi2abq.vbs** file is stored. By default, this location is **C:\Program Files\Autodesk\Moldflow Insight xxxx\data\commands**, (where **xxxx** is the version number of the Autodesk Moldflow Insight software you are using)
3. Click on the **mpi2abq.vbs** script, and then click **Open**.

   The script plays.

The API script saves the interface files in a folder created within the current project folder (for example in Windows XP, **My Documents\My AMI xxxx Projects\Project1\<study_name>_interface_files**, or in Windows Vista, **Documents\My AMI xxxx Projects\Project1\<study_name>_interface_files**, where **xxxx** is the version number of the Autodesk Moldflow Insight software you are using and *<study_name>* is the name of the study you are exporting to Abaqus).

Once the files are in that location, you can use them in Abaqus. Refer to your Abaqus manual for instructions on how to import them.

Exporting 3D data to Abaqus from a command line

NOTE: This feature is only available for 3D models on PC.

There are two parts to exporting Fill+Pack, Fiber and Warp results for 3D models to Abaqus: the first is running the **mpi2abq.vbs** macro to create the necessary files; the second is locating the files and using them in Abaqus.

To run the **mpi2abq.vbs** macro from the command line:

NOTE: The Autodesk Moldflow Insight command line will look for scripts stored in Windows XP, **My Documents\My AMI xxxx Projects\commands**, or in Windows Vista, **Documents\My AMI xxxx Projects\commands**. If no script is found, it will look in **C:\Program Files\Autodesk\Moldflow Insight xxxx\data\commands**, (where **xxxx** is the version number of the Autodesk Moldflow Insight software you are using).

---

1 Click **View tab > Windows panel > User Interface** and then select **Command Line**.

The **Command Line** dialog appears.

2 Type **mpi2abq**.

3 Click **Go**.

The script plays.

The API script saves the interface files in a folder created within the current project folder (for example in Windows XP, **My Documents\My AMI xxxx Projects\Project1\<study_name>_interface_files**, or in Windows Vista, **Documents\My AMI xxxx Projects\Project1\<study_name>_interface_files**, where **xxxx** is the version number of the Autodesk Moldflow Insight software you are using and *<study_name>* is the name of the study you are exporting to Abaqus).

Once the files are in that location, you can use them in Abaqus. Refer to your Abaqus manual for instructions on how to import them.

Autodesk Moldflow Insight to ANSYS interface

The Autodesk Moldflow Insight to ANSYS interface enables you to export the mesh and key material and analysis data for further stress analyses in ANSYS.

Injection molding specific distributions of initial stress, and material properties such as the orthotropic mechanical properties, and linear thermal expansion coefficients, are the key elements of this interface.

Prerequisites

The Autodesk Moldflow Insight to ANSYS interface requires the following:

- A Midplane or 3D meshed model

NOTE: A multi-cavity model is separated into individual cavities.

- Results of a Fill+Pack analysis, with or without Fiber orientation results

Restrictions

The following restrictions apply to the Autodesk Moldflow Insight to ANSYS interface:

- The following model features are not exported:
  - Runners, sprues, and gates
  - Cooling channels, bubblers, baffles
  - Mold boundary
  - Gas-assisted injection molding parts
  - Co-injection molding parts
- The following Autodesk Moldflow Insight simulation features are not considered in the ANSYS analysis:
  - Corner effects
  - Mold thermal expansion

Assumptions

When working with layered, spatially varying materials, the ANSYS Interface for Autodesk assumes the following:

- 20 laminates will be output.

  NOTE: Fill+Pack outputs 12 laminates by default and will therefore be interpolated to 20 laminates for ANSYS.

Data exported / files created

The Autodesk Moldflow Insight to ANSYS interface creates the following files:

- one *.cdb (command database) file
- one *.ist (initial stress) file

The data exported by the interface includes:

- For all Midplane/3D meshes:
  - Node position and element connectivity data
  - Material IDs and section IDs needed for element and laminated properties
  - Section data
- For a Midplane model and unfilled material:
  - Elastic modulus, shear modulus, and Poisson's ratio values from the material properties database

- Coefficient of thermal expansion (CTE) value from the material properties database
- Per-element and per-layer residual stress values in elemental local coordinate system
- Per-element and per-layer section orientation angle defined with respect to layer element coordinate system

- For a Midplane model and fiber-filled material:
  - Per-element and per-layer elastic moduli, shear moduli, and Poisson's ratio values in principal directions
  - Per-element and per-layer coefficient of thermal expansion (CTE) values in principal directions
  - Per-element and per-layer residual stress values in elemental local coordinate system
  - Per-element and per-layer section orientation angle defined with respect to layer element coordinate system

- For a 3D model and unfilled material:
  - Elastic modulus, shear modulus, and Poisson's ratio values from the material properties database
  - Coefficient of thermal expansion (CTE) value from the material properties database
  - Per-element initial stress values
  - If the unfilled material is transversely isotropic, the flow direction of each element will be the first principal direction

- For a 3D model and fiber-filled material:
  - Per-element and per-layer elastic modulus, shear modulus and Poissons ratio values in the three global coordinate directions
  - Per-element and per-layer coefficient of thermal expansion (CTE) values in the three global coordinate directions
  - Per-element initial stress values in the three global coordinate directions

Additional notes

The interface automatically creates a set of three fixities such that the nodes construct a maximum inscribed circle to fix the rigid body motion only for warpage calculation. You may wish to apply alternate displacement constraints within ANSYS prior to analysis.

ANSYS has stricter rules regarding element aspect ratio so the Autodesk Moldflow Insight mesh may generate a number of warnings within ANSYS. These warnings may be ignored.

For an Autodesk Moldflow Insight Midplane mesh, the interface creates *shell181* elements. If there are structural beams in the model, they are created as *beam4* elements. You can use a different element type that supports laminate initial stress and material property data by providing a suitable mapping in the *.cdb (command database) file generated by the interface.

For an Autodesk Moldflow Insight 3D mesh that comprises four-node tetrahedra, the interface creates 10-node *solid187* elements. This requires a mid-point generation command to be run in the ANSYS pre-processor. You can use a different solid element that supports orthotropic properties by providing a suitable mapping in the *.cdb file generated by the interface.

The ANSYS SHELL181 element includes the effects of transverse shear deformation. This requires the shear moduli G23 and G13 to be used in this element, which is different to what the Autodesk Moldflow Insight Warp solver does.

ANSYS cannot be automatically instructed to run a large deflection analysis; a linear analysis with one loading step is run by default. If you have selected a large deflection analysis in Autodesk Moldflow Insight, you need to make appropriate changes to the ANSYS command script to also run a large deflection analysis; otherwise, the Autodesk Moldflow Insight and ANSYS results will not be comparable.

Autodesk Moldflow Insight 3D Warp analysis is based on proprietary solver technology, and the element type(s) used in ANSYS can differ from those used in 3D Warp models. Therefore, there may be a difference in the magnitude of the deflections calculated by ANSYS 3D and 3D Warp analyses. However, the warped shape should be the same in both results.

Autodesk Moldflow Insight to ANSYS interface

The Autodesk Moldflow Insight to ANSYS interface enables you to export the mesh and key material and analysis data for further stress analyses in ANSYS.

Exporting model and results to ANSYS

NOTE: This feature is available only for Midplane or 3D models when an analysis sequence that includes Fill+Pack is selected.

There are two parts to exporting Fill+Pack results (with or without Fiber orientation results) from Autodesk Moldflow Insight to ANSYS. The first is running the **mpi2ans.vbs** script to create the necessary interface files; the second is locating these interface files and using them in ANSYS. The interface will convert any Midplane or 3D model from Autodesk Moldflow Insight into an ANSYS command database file. It will reject Dual Domain models because these cannot be taken into ANSYS for structural analysis.

To run the **mpi2ans.vbs** macro:

1. Open the Midplane or 3D study with Fill+Pack results.
2. Click **View tab > Windows panel > User Interface**, and select **Command Line**.
3. Type **mpi2ans** and click **Go**.

NOTE: If unused properties exist, a warning dialog appears informing you that they will be removed. Click **OK** to close this dialog.

The script will then prompt for an output filename.

TIP: To ensure that ANSYS will not have any difficulty reading the files generated by this interface, use only alphanumeric characters and underscores in the filename. In particular, you should avoid using parentheses: "(" ")".

4. Accept the default output filename or type an alternate name, and click **OK**.
5. In the **Setting units system** dialog, enter **SI**, **English** or **metric** to specify the units system, and click **OK**.
   A dialog appears confirming the selected units to be used. Click **OK**.
6. Click **OK** in the dialog that appears to confirm the name of the output folder.

   The script creates a folder of this name under the project directory of the study being exported, and then launches an executable in a DOS window to generate the ANSYS input files.

   NOTE: If a Windows Security Alert window appears, click **Unblock** to allow the executable *mpi2str* to run.

7. Click **OK** to confirm the export is complete.

   TIP: If you plan to compare Autodesk Moldflow Insight and ANSYS results, note down the numbers of the three nodes to which the interface applied constraints (the *anchor nodes*), as reported in the DOS window.

NOTE: If the model comprises multiple cavities, then a set of three constraints will be reported for each individual cavity. You will also find that the interface has created separate ANSYS input files (*.ist, *.cdb) for each cavity, with the number 1, 2, etc. appended to the output file name you specified.

Locate the files that have been generated. These files are stored in the Autodesk Moldflow Insight project directory where the interface files were created. Open the **moldflow2ansys.db** file in ANSYS and generate the required results using the ANSYS documentation for support.

CAUTION: When viewing Autodesk Moldflow Insight results for the purpose of comparison with the results you obtained from ANSYS, be sure to set an anchor plane based on the anchor nodes selected by the interface. Note also that the Autodesk Moldflow Insight analysis considers certain injection molding specific effects that are not simulated in ANSYS, namely corner effects and mold thermal expansion.

Mapping pressure or temperature results to a mold mesh surface for export to ANSYS for mold deflection analysis

To map pressure or temperature time series results to a mold mesh surface for analysis with ANSYS software, you must have separate 3D meshed models of the part and the mold, stored in separate study (***.sdy**) files in the same project.

- The mold mesh property should be defined as either **Part insert (3D)** or **Core (3D)**.
- The specified **Mold material** property must be the same on both the part model and the mold model.
- The part model and mold model geometries must mate to each other, but the nodes and elements on the mating surfaces do not have to coincide.
- The mold mesh must have enough nodal constraints applied for stress analysis.

This feature is available only for 3D models when an analysis sequence that includes Fill+Pack is selected.

NOTE: This feature is not supported for the Underfill Encapsulation molding process.

---

1 Ensure that you have 3D meshed models of the part and the mold saved in separate studies in the current project; for example, **part.sdy** and **mold.sdy**.

  Typically, these models are prepared for a core deflection analysis, and the mating surfaces must match.

2 Ensure that the mold properties specified for the mold mesh are identical to the properties that are defined for the mold steel in the part mesh.

3 Open the part study (for example, **part.sdy**) and select an analysis sequence that includes Fill+Pack.

4 Click  (**Home tab > Molding Process Setup panel > Process Settings**) to open the **Process Settings Wizard**.

  a If necessary, click **Next** until the page on which the **Advanced options** button appears is displayed.

    For thermoplastics molding processes, this is the **Fill+Pack Settings** page. For thermoset molding processes, this is the **Profile Settings** page.

  b Click **Advanced options**.

  c In the **Solver parameters** group, click **Edit.**

  d Click the **Interface** tab, and click **ANSYS options**.

5 In the **ANSYS Options** dialog, set the **Separate finite element mesh for mold** option to Specify it, and click **Specify filename**.

a In the **Mold mesh model filename** text box, type the name of the study in the current project that contains the mold mesh model, for example, **mold.sdy**.

b From the **Select part data to pass onto mold mesh** list, select the results you want to map onto the mold mesh: Pressure history, Temperature history, or Both pressure and temperature.

c From the **Solution method** list, select the type of analysis to perform in ANSYS: One step steady or Multi-step transient.

6. Click **OK** four times to return to the **Process Settings Wizard**.
7. Specify any remaining process settings, if necessary click **Next** until the last page appears, and click **Finish** to close the **Process Settings Wizard**.
8. Click  (**Home tab > Analysis panel > Start Analysis**).

When the analysis is complete, run the **mpi2ans.vbs** macro to generate the interface files for input to ANSYS. See *Exporting model and results to ANSYS* for details about generating the interface files.

Autodesk Moldflow Insight to LS-DYNA interface

The LS-DYNA Interface enables Autodesk Moldflow Insight users to output Midplane mesh models and some key material properties to LS-DYNA input files so that various analyses can be done with LS-DYNA.

NOTE: This feature is available only on Windows systems.

An API script, **mpi2dyn.vbs**, automatically converts the necessary Autodesk Moldflow Insight files into a format that LS-DYNA can use.

Injection molding-specific distributions resulting from a Warp analysis, such as initial stress and material properties, are the key elements of this interface. These distributions are written to separate LS-DYNA input files, including the **.sts* file for initial stress distribution and the **.mts* file for mechanical property distribution.

Release version 971 of LS-DYNA or later versions are required to work with this LS-DYNA Interface.

Interfacing to LS-DYNA

The LS-DYNA Interface passes three data types from Autodesk Moldflow Insight to LS-DYNA according to LS-DYNA input specifications. These data types are:

1. mesh model data,
2. initial stress data, and
3. material data.

Another type of data is the control parameter set. In the LS-DYNA Interface, control parameters are grouped into a default set for predicting the warped part shape. However, the control parameters will differ depending on the situation and the solution level of difficulty, so users may choose to change the default options.

To view or change the LS-DYNA options in Autodesk Moldflow Insight for the current study, ensure that you have selected an analysis sequence that includes Fill+Pack, click **Analysis > Process Settings Wizard** , if necessary click **Next** one or more times to navigate to the **Fill+Pack Settings** page of the Wizard, click **Advanced options**, click **Edit** in the Solver parameters group, click **Interface** (tab), and click **LS-Dyna options**.

Details about these options can be found in the LS-DYNA documentation; the default values are given in this **LS-DYNA Options** dialog.

Material model options

LS-DYNA includes more than 200 material models from which to choose. The LS-DYNA Interface includes four material model options considered most applicable to analysis of thermoplastics injection-molded parts.

- MAT_116 models the elastic responses of composite layups that have arbitrary layers through the part thickness and is based on the orthotropic assumption, so it is equivalent to the approach used in the Warp solver. This is the default material model option in the LS-DYNA Interface.
- MAT_022 is the composite damage model. However, the Autodesk Moldflow materials Database does not include polymer matrix and fiber strength properties.

  NOTE: If you choose this option, you will need to fill in the actual strength values for impact analysis in LS-DYNA.

- MAT_023 is the orthotropic temperature dependency model. However, the Autodesk Moldflow materials Database does not include relaxation moduli or a temperature dependency model for structural analysis.

  NOTE: Currently, only two sets of the same values corresponding to two temperatures are used in the LS-DYNA Interface. If necessary, you will need to replace the temperature dependent data and provide more temperature points in LS-DYNA.

- MAT_002 is the orthotropic elastic model. This is a simple model which should be used only with unfilled materials.

Solution method options

There are two basic solution method options available in LS-DYNA: static or quasi-static. In general, applications using unfilled materials can be solved easily by the static method, but for some applications using fiber-filled materials, the quasi-static method may be necessary. Users need to understand the time range and other control parameters associated with the quasi-static method in LS-DYNA.

Memory limit option

By default, the LS-DYNA Interface produces the LS-DYNA input file with a maximum of 300,000,000 words. However, interface files for typical Autodesk Moldflow Insight models of 6,000 elements can exceed this default value, such that the Windows 32-bit version of LS-DYNA may not be able to handle them. Typically, for models having more than 6,000 elements and using fiber-filled materials, the LS-DYNA analysis must be run on a UNIX or LINUX workstation. In this case, you can change the default memory value in the **LS-DYNA Options** dialog to a higher value, or you can change the *KEYWORD value in the ASCII input file.

Other options

Refer to the LS-DYNA documentation for information about other element formulation options and parallel processing options.

Converting LS-DYNA interface files for use on UNIX/LINUX systems

Since the LS-DYNA Interface is only available on Windows systems, whereas the LS-DYNA application itself is often run on UNIX or LINUX systems, it is possible to convert interface files generated on Windows systems for use on UNIX/LINUX systems: in the C shell, type **tr -d** '**\015**' < **filename.dyn** > **tmp**, then type **mv tmp filename.dyn**.

Exporting to LS-DYNA

There are two ways to export data to LS-DYNA: from the user interface, or from the command line.

Exporting data to LS-DYNA from the user interface

NOTE: This feature is only available on Windows systems.

The Autodesk Moldflow Insight to LS-DYNA interface requires the following:

- A Midplane meshed model.
- Results of a Fill+Pack + Warp analysis sequence, with Fiber results if a fiber-filled material is selected.

  NOTE: Set the desired LS-DYNA interface options before launching the analysis.

- A license to use the LS-DYNA Interface.

There are two parts to exporting Autodesk Moldflow Insight data for input to LS-DYNA: running the **mpi2dyn.vbs** macro to create the necessary files, and locating the files and using them in LS-DYNA.

To run the **mpi2dyn.vbs** macro from the user interface:

1. Click **Tools tab > Automation panel > Play Macro**.
2. In the **Open Macro** dialog, navigate to the location where the **mpi2dyn.vbs** file is stored. By default, this location is **C:\Program Files\Autodesk\Moldflow Insight xxxx\data\commands**, where xxxx is the version number of the Autodesk Moldflow Insight software you are using.
3. Click on the **mpi2dyn.vbs** script, and then click **Open**.

   The script plays.

The API script saves the interface files in a folder created within the current project folder. For example in Windows XP, **My Documents\My AMI xxxx Projects\Project1\<study_name>_interface_files**, or in Windows Vista, **Documents\My AMI xxxx Projects\Project1\<study_name>_interface_files**, where xxxx is the version number of the Autodesk Moldflow Insight software you are using and **<study_name>** is the name of the study you are exporting to LS-DYNA.

Once the files are in that location, you can use them in LS-DYNA. Refer to your LS-DYNA manual or help system for instructions on how to import them and perform the LS-DYNA analysis.

Exporting data to LS-DYNA from the command line

NOTE: This feature is only available on Windows systems.

The Autodesk Moldflow Insight to LS-DYNA interface requires the following:

- A Midplane meshed model.
- Results of a Fill+Pack + Warp analysis sequence, with Fiber results if a fiber-filled material is selected.

  NOTE: Set the desired LS-DYNA interface options before launching the analysis.

- A license to use the LS-DYNA Interface.

There are two parts to exporting Autodesk Moldflow Insight data for input to LS-DYNA: running the **mpi2dyn.vbs** macro to create the necessary files, and locating the files and using them in LS-DYNA.

To run the **mpi2dyn.vbs** macro from the command line:

NOTE: The Autodesk Moldflow Insight command line will look for scripts stored in Windows XP, **My Documents\My AMI xxxx Projects\commands**, or in Windows Vista, **Documents\My AMI xxxx Projects\commands**. If no script is found, it will look in **C:\Program Files\Autodesk\Moldflow Insight xxxx\data\commands**, (where **xxxx** is the version number of the Autodesk Moldflow Insight software you are using).

---

1. Click **View tab > Windows panel > User Interface** and then select **Command Line**.
2. Enter **mpi2dyn**.
3. Click **Go**.

   The script plays.

The API script saves the interface files in a folder created within the current project folder. For example, in Windows XP, **My Documents\My AMI xxxx Projects\Project1\<study_name>_interface_files**, or in Windows Vista, **Documents\My AMI xxxx Projects\Project1\<study_name>_interface_files**, where xxxx is the version number of the Autodesk Moldflow Insight software you are using and *<study_name>* is the name of the study you are exporting to LS-DYNA.

Once the files are in that location, you can use them in LS-DYNA. Refer to your LS-DYNA manual or help system for instructions on how to import them and perform the LS-DYNA analysis.

Exporting to PATRAN

There are two parts to exporting Fill+Pack, Fiber and Warp results from Autodesk Moldflow Insight to PATRAN. The first is running the **mpi2pat.vbs** macro to create the necessary files; the second is locating the files and using them in PATRAN. The interface will convert any Autodesk Moldflow Insight Midplane or 3D model to a PATRAN command database file.

NOTE: This feature is only available on Windows platforms.

A prerequisite of using the Autodesk Moldflow Insight to PATRAN interface is to run Fill+Pack analysis, with or without fiber orientation analysis first.

For a 3D model, 3D Warp needs to run to produce the initial stress values to be converted into PATRAN interface files. The 3D initial stress values are first saved in the *.tsp file in the project folder. After running the **mpi2pat.vbs** macro, they are saved in the <filename>_initStress.ele file in Patran format.

The mesh model will be converted to the PATRAN 2.5 Neutral file format (*.pat). The result data will be converted to PATRAN 2.5 Results Files (*.ele), and each layer has its own file with extension *.ele.008, corresponding to the layer number.

NOTE: Exporting to 3rd-party CAE formats requires that you have purchased the correct licenses. Refer to the Minimum license requirements page to find out if you can perform this operation.

Exporting to PATRAN

A prerequisite of using the Autodesk Moldflow Insight to PATRAN interface is to run Fill+Pack analysis, with or without fiber orientation analysis first.

Exporting to PATRAN

Exporting to 3rd party CAE formats requires that you have purchased the correct licenses. Refer to the Minimum license requirements page to find out if you can perform this operation.

To run the **mpi2pat.vbs** macro:

1. Open the Midplane or 3D study with Fill+Pack results, with or without fiber orientation.

   NOTE: Dual Domain models will be rejected as these cannot be taken into PATRAN for structural analysis.

2. Click **View tab > Windows panel > User Interface** and then select **Command Line**.
3. Type **mpi2pat** and click **Go**.

   The script will first prompt for an output filename.

   NOTE: To ensure that PATRAN will not have any difficulty reading the files generated by this interface, use only alphanumeric characters and underscores in the filename. In particular, you should avoid using parentheses: "(", ")".

4. Accept the default output filename or enter an alternate name, and then click **OK**.
5. Click **OK** to confirm the message informing you of the name of the output folder.

   The script creates a folder of this name under the project directory of the study being exported, and then launches an executable in a DOS window to generate the PATRAN input files.

   NOTE: If a Windows Security Alert window appears, click **Unblock** to allow the executable to run.

6. Click **OK** to confirm the message that the export is complete.

The API script saves the interface files in a folder created within the current project folder. For example, in Windows XP, **My Documents\My AMI xxxx Projects\Project1\<study_name>_interface_files**,

or in Windows Vista,
Documents\My AMI xxxx Projects\Project1\<study_name>_interface_files, where **xxxx** is the version number of the Autodesk Moldflow Insight software you are using and <*study_name*> is the name of the study you are exporting to PATRAN.

Once the files are in that location, you can use them in PATRAN. Refer to your PATRAN manual for instructions on how to import them.

Export to NASTRAN

There are two parts to exporting Fill+Pack, Fiber and Warp results from Autodesk Moldflow Insight to NASTRAN. The first is running the **mpi2nas.vbs** macro to create the necessary files; the second is locating the files and using them in NASTRAN. The interface will convert any Autodesk Moldflow Insight Midplane or 3D model to a NASTRAN command database file.

NOTE: This feature is only available on PC.

A prerequisite of using the Autodesk Moldflow Insight to NASTRAN interface is to run Fill+Pack analysis, with or without fiber orientation, first.

For a 3D model, 3D Warp needs to run to produce the initial stress values to be converted into NASTRAN interface files. The 3D initial stress values are first saved in the *.tsp file in the project folder. After running the **mpi2nas.vbs** macro, they are converted to initial strains and saved in the *.ist file that NASTRAN can read in.

The mesh model and related material properties and initial stresses are converted to the NASTRAN Bulk Data Format, primarily using the free form in order to reduce the interface file size. The interface file extensions used are *.nas for the mesh model, *.mts for the material properties, and *.ist for the initial strains.

Export to NASTRAN

A prerequisite of using the Autodesk Moldflow Insight to NASTRAN interface is to run Fill+Pack analysis, with or without fiber orientation, first.

Export to NASTRAN

To run the **mpi2nas.vbs** macro:

1. Open the Midplane or 3D study with Fill+Pack results, with or without fiber orientation.

   NOTE: Exporting to 3rd party CAE formats requires that you have purchased the correct licenses. Refer to the Minimum license requirements page to find out if you can perform this operation.

   ---

2 Click **View tab > Windows panel > User Interface** and then select **Command Line**.

3 Type **mpi2nas** and click **Go**.

The script will first prompt for an output filename.

4 Accept the default output filename, or enter an alternate name, and then click **OK**.

5 Click **OK** to confirm the message informing you of the name of the output folder.

The script creates a folder of this name under the project directory of the study being exported, then launches an executable in a DOS window to generate the NASTRAN input files.

NOTE: If a Windows Security Alert window appears, click **Unblock** to allow the executable to run.

---

6 Click **OK** to confirm the message that the export is complete.

The API script saves the interface files in a folder created within the current project folder. For example, in Windows XP, **My Documents\My AMI xxxx Projects \Project1\<study_name>_interface_files**, or in Windows Vista, **Documents\My AMI xxxx Projects \Project1\<study_name>_interface_files**, where **xxxx** is the version number of the Autodesk Moldflow Insight software you are using and **<study_name>** is the name of the study you are exporting to NASTRAN.

Once the files are in that location, you can use them in NASTRAN. Refer to your NASTRAN manual for instructions on how to import them.

Export to Altair Hyper3D

You can export the model and analysis result data from the active study to an Altair Hyper3D (*.h3d) file. This format is directly supported by Altair Engineering visualization products.

IMPORTANT: HyperView 9.0 displays mesh and result data associated with mesh nodes or elements. HyperView Player 9.0 displays mesh only. Earlier versions of HyperView and HyperView Player cannot display exported *.h3d files.

NOTE: This feature is supported for Midplane, Dual Domain, and 3D analysis technologies.

- The model mesh and available result data are saved in the *.h3d file.
- If no results are available in the study, only the mesh is included in the exported file.
- If results are available and no results are marked for export, all results will be included with the mesh in the exported file.

- If you have marked results for export, only the marked results will be included with the mesh in the exported file.

NOTE: The *.h3d format only supports the export of result data that is associated with mesh nodes and elements. Other result data, such as Molding Window analysis results and X-Y plot data, are not supported.

Export to Altair Hyper3D

You can export the model and analysis result data from the active study to an Altair Hyper3D (*.h3d) file.

Exporting an Altair Hyper3D file

You can export the model and analysis result data from the active study to an Altair Hyper3D (*.h3d) file. This format is directly supported by Altair Engineering's visualization products.

IMPORTANT: HyperView 9.0 displays mesh and result data associated with mesh nodes or elements. HyperView Player 9.0 displays mesh only. Earlier versions of HyperView and HyperView Player cannot display exported *.h3d files.

NOTE: This feature is supported for Midplane, Dual Domain, and 3D analysis technologies.

---

1. Click **Results tab > Export and Publish panel > Moldflow Results**.

   Alternatively, click then **Export > Study & Results** .

   The **Export** dialog opens.
2. Navigate to the location in which you want to save the export file.
3. In the **File name** text box, enter a name for the export file.
4. Select **Altair Hyper3D (*.h3d)** in the **Save as type** drop-down list, and then click **Save**.

The model mesh and available result data are saved in the *.h3d file. If no results are available in the study, only the mesh is included in the exported file. If results are available and no results are marked for export, all results will be included with the mesh in the exported file. If you have marked results for export, only the marked results will be included with the mesh in the exported file.

CAUTION: The *.h3d format only supports export of result data associated with mesh nodes and elements. Result data such as Molding Window analysis results and X-Y plot data are not supported.

Export to Code V

Code V is an optical analysis program from Optical Research Associates, which can simulate the appearance of an image seen through a lens.

You can export the optical properties of a lens analyzed with a Fill + Pack + Warp analysis to Code V, and investigate how the molding process affects the optical properties of the lens.

Export to Code V

You can export the optical properties of a lens analyzed with a Fill + Pack + Warp analysis to Code V, and investigate how the molding process affects the optical properties of the lens.

Creating a birefringence result for export to Code V

The Code V export script requires a Fill + Pack + Warp analysis with birefringence results.

NOTE: Check the Phase shift result for the magnitude of the retardance. Best results are obtained if the retardance in the optical area of the part is less than 5–10 wavelengths.

TIP: Orient the lens so that the principal optical direction is the Z axis. The +Z side of the lens should be the top of the lens. If the lens is oriented differently, create a local coordinate system that satisfies the same requirement.

Create a custom birefringence Retardance Tensor plot. This plot will be used by the export script to produce interface files that are imported into Code V.

1. Select **Results tab > Plots panel > New Plot > Custom**.
2. Select the Birefringence plot type.
3. Select the +Z direction.

   If you defined a local coordinate system, select it in the **Coordinate System** drop-down list.
4. Select **Retardance tensor**.

A new result Retardance tensor_+Z is created.

TIP: It is recommended that you define an anchor plane so that the optical axis is explicit. Without an anchor plane, the optical axis is computed using a “best fit” method.

Running the Code V interface script

The **mpi2codev.vbs** script is located in the **data\commands** directory where Autodesk Moldflow Insight was installed.

1 Click **View tab > Windows panel > User Interface** and then select **Command Line**.

2 Type **mpi2codev** and press **Go**.
The **Enter filename** dialog appears.

3 Enter a prefix that will become the base name of the files that the script generates.

   If your prefix contains an absolute path, all of the generated files will be placed in that location.

4 Enter the number of grid points in the X and Y directions.

   This controls the resolution of the generated files. Values between 10 and 50 are usually appropriate.

   CAUTION: A very coarse grid will reduce the accuracy of the Code V analysis. A very fine grid may result in too few mesh elements contributing to each grid point, magnifying rounding errors and affecting the accuracy. The script warns you if the mesh is too coarse with respect to the grid.

5 Enter the minimum and maximum X and Y coordinates corresponding to the optical area of the lens.

   Coordinates are in the same units as the study (for example, millimeters or inches).

The script creates four files, each with the same base name as the prefix you chose.

Table 2: Interface files created by mpi2codev.vbs

File name	Description
***prefix*_Top_SUR.int**	Warpage of the top surface
***prefix*_Bottom_SUR.int**	Warpage of the bottom surface
***prefix*_BIR.int**	Average birefringence of the part **NOTE:** This interface file corresponds to the Autodesk Moldflow Insight Phase shift result, but will look different when plotted in Code V because the result is displayed relative to the wavelength in Code V.
***prefix*_CAO.int**	Crystal axis data for the part

Importing the interferogram files into Code V

Code V considers the top and bottom surfaces of a lens as independent entities. You must attach the generated surface interface files to the corresponding surface in Code V. Attach the birefringence interface files to the top surface in Code V.

NOTE: Meniscus lenses with a bevel effectively have a smaller diameter for the concave surface than for the convex surface. You may need to run the **mpi2codev.vbs** script twice, specifying the diameters for the top and bottom surfaces, and import the corresponding version into Code V on each surface.

---

1. Edit the surface properties of the top surface.

   NOTE: In order to be attached to one surface, the three interferogram files must be given different labels in Code V.

   ---

   a. Attach the ***prefix*_Top_SUR.int** interferogram file.
   b. Attach the ***prefix*_CAO.int** interferogram file.
   c. Attach the ***prefix*_BIR.int** interferogram file.
2. Edit the surface properties of the bottom surface.
   a. Attach the ***prefix*_Bottom_SUR.int** interferogram file.

If you are analyzing a multi-lens assembly, repeat the process for each lens.

MPX 3

You can simulate actual injection molding machine characteristics by importing data about the machine from Moldflow Plastics Xpert (MPX).

MPX

You can simulate actual injection molding machine characteristics by importing data about the machine from Moldflow Plastics Xpert (MPX).

Importing machine characteristics from MPX

The machine data must be exported from MPX in UDM file (ASCII model file *.udm) format. The data is incorporated into the current study.

NOTE: Importing machine characteristics is only available for the Thermoplastic Injection Molding Process.

1. Open the project and study for which you want to import data.
2. Click **(Home tab > Import panel > Import)**.
3. Locate the **UDM File (*.udm)** that contains the data you want to import.
4. Click **Open**.

The relevant machine information has been transferred to the current study and can be reviewed in the Process Settings Wizard.

Editing imported machine characteristics

You can edit or verify machine characteristics that have been imported into the current study from MPX.

NOTE: Importing machine characteristics is only available for the Thermoplastic Injection Molding process.

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

   The **Fill+Pack Analysis Advanced Options** dialog appears.
3. Click **Edit...** next to the **Injection molding machine** feature.

4 Select the appropriate tab from the top of the dialog and edit the machine parameters as required.
5 Click **OK** to accept the data, and then close all other dialogs.

Importing process settings from MPX

You can simulate actual injection molding process conditions by importing data about these conditions from Moldflow Plastics Xpert (MPX).

The process conditions must be exported from MPX in UDM file (ASCII model file *.udm) format. The data is incorporated into the current study.

NOTE: Importing processing conditions is only available for the Thermoplastic Injection Molding Process.

---

1 Open the project and study for which you want to import data.
2 Click **Home tab > Molding Process Setup panel > Import MPX Process Settings**.
3 Locate the **UDM File (*.udm)** that contains the data you want to import.
4 Click **Open**.

The relevant process conditions have been transferred to the current study and can be reviewed in the Process Settings Wizard.

Editing imported process settings

You can edit or verify process settings that have been imported into the current study from MPX.

NOTE: Importing processing conditions is only available for the Thermoplastic Injection Molding process.

---

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

   The **Fill + Pack Analysis Advanced Options** dialog appears.
3 Click **Edit...** in the **Process controller** pane.
   The **Process controller** dialog appears.
4 Click the **MPX Profile Data** tab and click **Edit profile...**.
   The **Measured/Fitted Profile Data from** MPX dialog appears.
5 Select and edit the required data.

   NOTE: The stroke must be entered in ascending order.

   ---

6 Click **OK** to accept the data, and then close all other dialogs.

Editing DOE settings after importing process variations

Ensure a **Design of Experiments (Fill + Pack)** analysis is selected as the **Analysis Sequence** in the **Study Tasks** pane.

You can edit or verify DOE settings after importing process variations data from MPX.

NOTE: Importing process variations is only available for the Thermoplastic Injection Molding process.

1 Click **Home tab > Molding Process Setup panel > Process Settings**.
2 Click **Next** to display the **DOE settings** page.
3 Adjust the DOE parameters as required.

   NOTE: When the option in a drop-down box is altered to **Specified**, you can adjust the range of values to be considered in your DOE by adjusting the value in the **Delta** text-box.

4 Click **Finish** to accept the DOE settings using the imported data.

MPX

Use this dialog to import data from MPX.

Measured/Fitted Profile Data from MPX dialog

This dialog shows the raw profile data imported using the **Analysis > Import Data From MPX > Import Process Settings** menu item, as well as curve-fitted data.

The measured or fitted data can also be viewed graphically by clicking on the corresponding **Plot** button.