



Purging Old Misconceptions: Best Practices for Meshing in Simulation Moldflow Insight

Caitlin Tschappat – Autodesk, Inc.

Justin Courter – Autodesk, Inc.

SM5706-P

This class will cover the use of meshing options in Simulation Moldflow software. We frequently find that the Simulation Moldflow software user is unaware or unsure of which mesh type to use or what it is that constitutes a suitable mesh for a given geometry. We will discuss what a suitable mesh consists of and how the mesh helps to create more accurate results when running an analysis. We will cover, among other things, the importance of a refined mesh versus a course mesh, and we will discuss the instances when the over refinement of a mesh versus the under refinement no longer poses a benefit to an analysis. We will also explore common issues that we find with different mesh types, and we'll look at ways in which to manually improve the quality of a mesh.

Learning Objectives

At the end of this class, you will be able to:

- Understand the need for proper mesh representation and configuration
- Discover the influence of improper mesh use
- Discover the ideal use of mesh types, mesh options, and mesh configuration
- Learn how to incorporate other tools to improve quality

About the Speakers

Caitlin Tschappat graduated from Pennsylvania State University with a Bachelor of Science degree in Plastics Engineering Technology. Prior to working for Autodesk, Inc., she worked as an application engineer for Zurn Engineered Water Solutions, a plastic drainage company in Erie, Pennsylvania. She also worked for General Motors Company as a Production Group leader in Michigan. At Autodesk, Inc., Caitlin is a Frontline technical support specialist primarily focusing on Simulation Moldflow software. Caitlin gained her knowledge through hands-on experiences during her work at Penn State and through previous industry experiences. Recently Caitlin spoke at Autodesk Technical Academy in Denver, Colorado, where she delivered a presentation on general troubleshooting of challenges that users frequently encounter. This is Caitlin's first time presenting at Autodesk University.

Justin Courter is a Premium Support Specialist for Autodesk® Simulation® products, with a focus on Moldflow®, which he's used for over 10 years. In addition to his experience, Justin has technical certifications in both Simulation Moldflow and Simulation CFD products. His current role includes working with Enterprise Priority customers for Autodesk, helping deliver on initiatives that will benefit both the customer's strategic goals and the partnership they have with Autodesk. Justin completed his Bachelors of Science degree in Plastic Engineering in 2007 at Penn State. Justin has been a member of the Society of Plastics Engineers (SPE) for eight years and has presented published work at the SPE Annual Technical Conference (ANTEC™) in the field of stress-induced chemical resistance of polymer materials. He has also presented at Autodesk University in the past.

Understand the Need for Proper Mesh Representation and Configuration

Overview

The analysis mesh is a collection of elements and nodes that form the basis of the computations that are performed during the simulation. It is essential that a high quality mesh is produced from the CAD representation in order to perform simulation.

Mesh Importance

Mesh type and refinement has a significant impact on the results in Simulation Moldflow. This isn't restricted to just accuracy, although that is often the most impactful influence. It also has an influence on display and thus results display and availability, accurate representation of the geometry, and an influence on time to analyze.

Because of how Moldflow handles different element types based on the information they represent, the solver behavior can vary based on the mesh used. This primarily has to do with handling the geometry in an accurate way, to properly capture the behavior of all aspects of simulation. We'll cover this importance in much more detail on many of these very issues.

Creating a Mesh

We offer a variety of means with which you can generate, troubleshoot, and improve your part mesh. This flexibility allows users a variety of tools to ensure the best possible representation of the part to ensure the requisite level of accuracy. This includes best practices which require the use of outside tools, as well as a means to get a suitable mesh using only Simulation Moldflow.

We'll detail many of these different options, as well as provide a Autodesk recommended workflow.

Discover the Influence of Improper Mesh Use

Overview

Improperly using mesh can have a variety of influences on the results with in Moldflow, and the severity of such results can also range. It can be something as simple as not having access to a specific result only available in one mesh type, or it can be the complete invalidation of the accuracy of your data. This of course is because mesh is the representation of the part, and a poor quality mesh can lead to poor geometric representation which can lead to incorrect calculations and diminished accuracy.

This section will detail that influence to help provide context to why these best practices are important, as well as highlight what many of those best practices actually are.

Model Quality

To begin, it's worth looking at what is often the very first step in setting up a Moldflow model. This is the source data provided; specifically we'll be focusing on the part geometry and how it is represented.

Types of Files Handled by Simulation Moldflow Insight

Before we get into anything else, it's important to know exactly what file types can be used in Simulation Moldflow. Via integrated plug-ins, there are a variety of file formats that are usable within Moldflow in some capacity.

File format	Recognized file extensions	Direct Import of native CAD format
Moldflow Study File	*.sdy	N/A
ANSYS Prep 7	*.ans	N/A
I-DEAS Universal	*.unv	N/A
NASTRAN Bulk Data	*.bdf	N/A
PATRAN Neutral	*.pat, *.out	N/A
Stereolithography	*.stl	N/A
ASCII model	*.udm	N/A
Autodesk Inventor	*.ipt	Yes
	*.iam	
Alias (v10 or later)	*.wire	No
Autodesk Shape Manager	*.smt, *.smb	Yes
CATIA® V5 (R6-R23)	*.catpart,	Yes
CATIA® V5 (R6-R23) Assembly	*.catproduct	
UGS NX 9	*.prt	Yes
Pro/ENGINEER® Wildfire® 5.0	*.prt,	Yes
Pro/ENGINEER® Wildfire® 5.0 Assembly	*.asm	
Creo™ Elements/Pro™ 5.0	*.prt,	Yes
Creo™ Parametric 2.0	*.prt,	Yes
Parasolid® V26	*.x_t, *.x_b, *.xmb, *.xmt	Yes
Rhino 5.0	*.3dm	Yes
SolidWorks® 2014	*.sldprt,	Yes
SolidWorks® 2014 Assembly	*.sldasm	
SAT (v4 - v7)	*.sat	Yes
STEP (AP214, AP203E2)	*.stp, *.step	Yes
IGES	*.igs, *.iges	No

Best Practices for Model Use

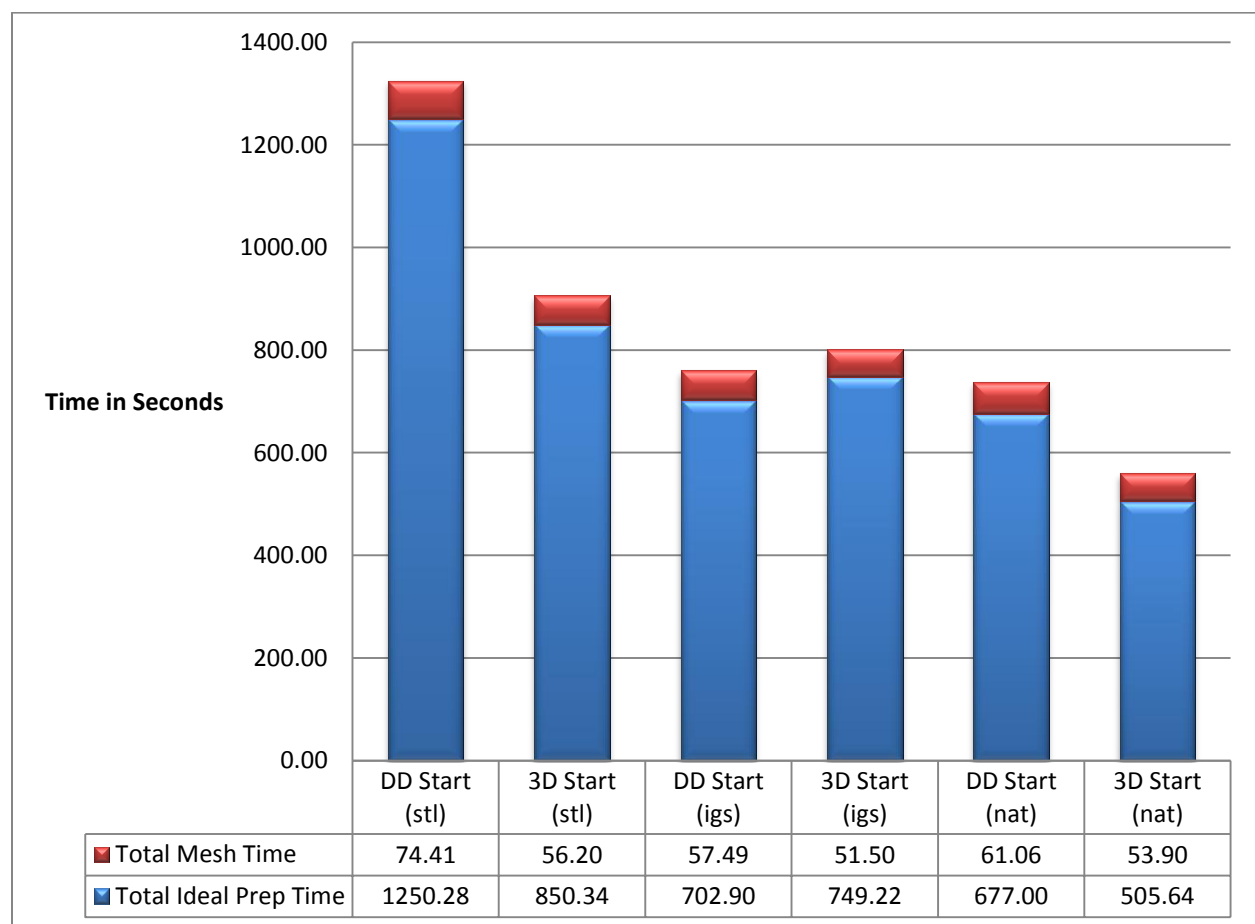
The ideal and recommended situation here is actually to use a native representation of the model. The primary reason being it is the most uncompromised representation. There is no modification of the geometry to another format, stitching of surfaces or generation of facets. And with the improvements to Moldflow Design Link, which is now automatically installed and licensed through Synergy licenses, we can now handle the native CAD kernels for many different file formats, as noted in the chart above. To add to that, the more recent the source

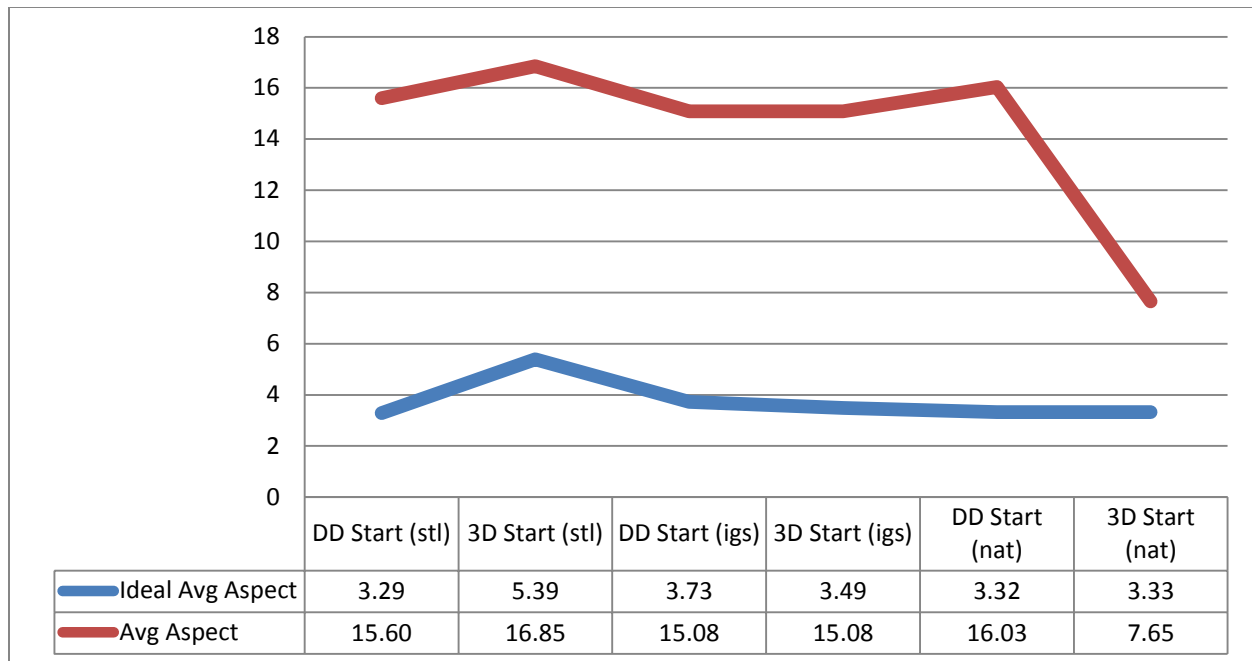
CAD program and file, the better as it is more likely to be tuned to the version of Simulation Moldflow you're using.

Of course there are often cases in which having access to the source data is not possible. The reasons for this are numerous, but unfortunately not uncommon. When using neutral files, such as IGES or STL the best approach is to ensure quality via a different medium than the CAD program in question. Often "fixing" a neutral file in a CAD program has little bearing on how Moldflow will read that file. It's suggested neutral files are run through CAD Doctor when possible, and CAD Doctor is covered in more depth in section four of this handout.

Influences on Results, Process and Time

The best way to highlight the importance of model quality is to show the impact it has on not just your results, but also the processes you need to follow as well as how long it will take to get your model setup and running. We've compiled some of the data shown in the presentation here in the handout to highlight the importance the quality of the source data can have on the other Simulation Moldflow process. In these tests a native file was used (SLDPRT) as well as two different neutral files of the same part (IGES and STL).





The first chart highlights the difference in both time to mesh (computation time) as well as total time to prepare the model for analysis. The second chart highlights the impact on a vital mesh statistic, average aspect ratio. The chart highlights the influence the model quality had on the original mesh with no manual modification, and the ideal mesh with manual improvement to the original mesh.

When it comes to the influence on the analyses and results, it's difficult to capture in a succinct way the level of influence due to the relevance of how poorly the quality of the file represents the geometry. The level of impact can be directly influenced by both how substantial the misrepresentation is as well as how sensitive the part, material, processing is to such misrepresentation. As such, no chart provide the level of detail necessary to show this concern.

We also need to consider the influence model size plays with respect to refinement. The time it takes to compute an analysis depends on several factors such as the size of the mesh, the number of elements, the number of nodes, and the number of layers. Adjusting the mesh can affect the amount of time it takes to run the analysis. Understanding the results trying to be achieved in reality will significantly help to understand how much mesh refinement is needed for a model.

Mesh refinement is an important step when creating the mesh for an analysis. It is important to understand the recommendations by Moldflow as well as why these guidelines are being recommended. Mesh refinement is important because it helps predict flow front patterns and imbalances that may occur as it provides a better representation of thermal characteristics. Mesh refinement also helps better predict results such as air traps, hesitation, race tracking, and weld lines. By using the mesh refinement guidelines, more desirable results are achievable.

Discover the Ideal Use of Mesh Types, Mesh Options, and Mesh Configuration

Overview

We've seen that misrepresenting the geometry or not following best practices (native CAD part) can have an impact on everything from the time to setup to the time to analyze to the impact on accuracy. When we look at understanding the meshing process, we need to approach it from two specific vantages. First we need to know what the general workflow or recommendations actually are. Secondly we need to apply context to why. Specifically why these are the recommendations, as well as a more complete understanding of each area with respect to meshing. That is how this section will be laid out, covering the process then the how's and why's of that process

Autodesk Recommended Workflow (Dual Domain and 3D)

Note: Some steps are relevant to need (i.e. no need to mesh to 3D if you're not running 3D or no need to generate a feed system if you aren't incorporating it in your analysis)

1. Acquire native CAD file
2. Check CAD defects in CADDr or CAD systems
 - a. Fix defects if found
3. Import the fixed CAD model into Moldflow (including inserts if part of assembly)
4. Check features closely and determine what should be the global edge length and where to use local mesh density
 - a. GEL to ensure at least 2 elements across thickness
 - b. GEL to ensure at least 3 elements to define width/height/length of features
 - c. Local mesh density to accurately capture specific areas
 - i. Complex geometries
 - ii. Need for higher results refinement
5. Generate a Dual Domain mesh (including inserts if part of assembly)
6. Determine mesh type to be used and analysis to be run
 - a. If running 3D analyses determine desired number of layers
 - i. Analysis type influences requirements
 1. 8 layers for fiber
 2. 10 layers for gas assist
7. Visibly survey to determine if mesh density is ideal (based on requirements)
 - a. If not, re-generate mesh with tightened parameters
8. Verify Mesh Statistics
 - a. Fix free edges, overlaps/intersections and triangles with aspect ratio > 20
 - b. Certain analyses require specific focus
 - i. Check Closeness if Cool (BEM) is needed. Fix manually if found
9. Mesh to 3D (if 3D analysis, including inserts if part of assembly)
10. Determine feed system representation
 - a. Beam for Dual Domain
 - b. Beam or 3D for 3D part
 - i. Based on feed system layout (3D for multiple transitions, complex layouts)
11. Add feed system geometry
 - a. Beams

- i. Import IGS curve layout
 - b. 3D
 - i. Import CAD representation
 - ii. Fix CAD defects per same process as part
- 12. Mesh feed system
 - a. Beams
 - i. Ensure GEL to capture at least 2 distinct elements per section
 - ii. Local mesh density tool ideal for increasing refinement towards the gate
 - b. 3D
 - i. Refer to steps 4-9 (Ensure contact surface between gate and part is connected)
- 13. Determine cooling analysis type
 - a. BEM, FEM or FEM Conformal
- 14. Import cooling system
 - a. BEM and standard FEM
 - i. Import IGS curve layout
 - b. Conformal
 - i. Import CAD representation (Ensure no issues as CAD model can't be fixed in Insight)
- 15. Mesh cooling system
 - a. Beam
 - i. Ensure GEL to capture at least 3 distinct element per section
 - ii. Maintain 2.5:1 length to diameter ratio
 - b. 3D
 - i. Define cooling inlet(s) and outlet(s)
 - ii. Mesh using specific cooling channel mesh tool
- 16. Import insert geometry if not imported as assembly of part or mold
 - a. import CAD representation
 - b. Fix CAD defects per same process for part
- 17. Mesh insert geometry
 - a. Refer to steps 8-12 (note for 3D internal refinement can be less than 6 layers)
- 18. Define insert property type based on end function
- 19. Determine necessary type of mold block representation
 - a. BEM cool
 - i. Boundary representation
 - b. FEM or FEM conformal
 - i. 3D mold block representation
- 20. Represent mold block
 - a. Boundary representation
 - i. Mold boundary representation tool within Moldflow to define dimensions and location
 - b. 3D mold block representation
 - i. Import CAD model of mold block
- 21. Mesh mold block
 - a. Boundary representation – N/A
 - b. 3D mold block
 - 1. Use 3D Mold mesh tool

Mesh Type Determination

Midplane

Midplane is the oldest technology Moldflow offers in the sense of meshing. Midplane mesh is a 2.5D representation of a solid model which is typically generated from the center line or mid-plane of the part. Generally speaking, it is more suitable for simple, thin-walled geometries.

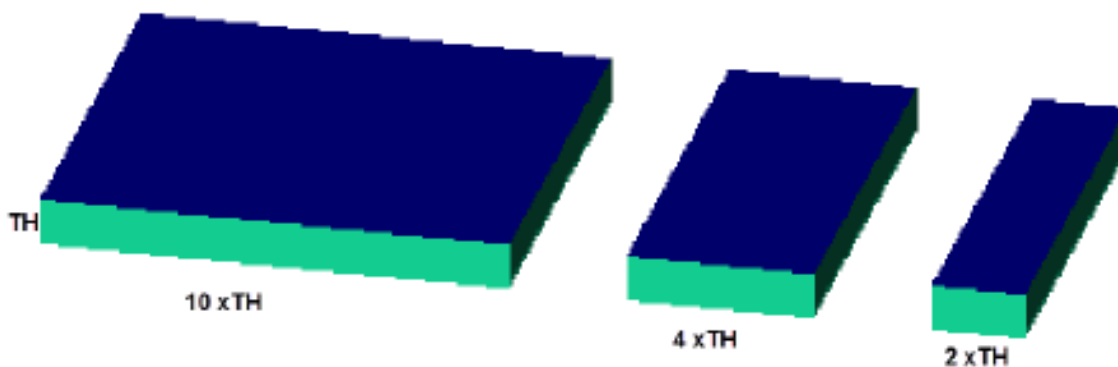
When using Midplane mesh, it is very important to have each element assigned a specific thickness. This is extremely crucial for predicting accurate flow as well as pressure predictions. A benefit to using this mesh type is that it is often easy to configure and offers the largest compilation of different analyses types available in Moldflow.

Dual Domain

The Dual Domain meshing technology is a unique and patented approach that obviates the need for Midplane generation for thin-walled plastic parts. The Dual Domain mesh is a shell like geometry, in which thicknesses are automatically generated from the geometric separation between the meshes on the two opposed surfaces of a Dual Domain mesh. For this reason, the mesh match and reciprocal mesh is a crucial mesh statistic in Moldflow Dual Domain meshing.

The Dual Domain mesh needs to have its elements aligned to another element on the opposite side to have the most accurate results. For this reason, Dual Domain is commonly recommended for thin walled applications as it is much easier to create suitable mesh match percentages and thus expected results.

The most significant reason why Dual Domain mesh type is preferred for thin-walled geometries is the inability to capture the heat transfer through the part when there is a complex, thick, chunky part. The heat transfer is not calculated in edge elements, and depending on the thickness to width ratio of the part, thick chunkier parts have a larger percentage of surface area excluded from the heat transfer calculation. In Moldflow, when determining the feasibility of using Dual Domain it is recommended the ratio to be no less than a 4:1. However, a ratio of greater than 10:1 is preferred.



3D

3D mesh type is the newest technology for Moldflow and is the primary focus moving forward. A 3D mesh uses tetrahedral and beam elements to represent the geometry. The biggest issue currently with 3D mesh type is in the creation of a mesh for thin walled parts. In order to appropriately capture the flow through the thickness, Moldflow has a minimum recommendation of six layers through the thickness for a flow analysis. With thin walled parts, this often requires re-evaluation of the surface mesh refinement to ensure a quality representation of the internal mesh. Because of this, the major downside to 3D is that it may not be as computationally efficient as Midplane or Dual Domain for these types of parts.

There are two ways users can create a 3D mesh in Moldflow. The user can import the CAD geometry and either mesh directly to 3D or mesh to Dual Domain first, then to 3D. Ideally you will create a Dual Domain mesh first, as it will allow for easier mesh improvements, via manual troubleshooting or remeshing. To convert to 3D you will require a water tight Dual Domain model. If you mesh directly to 3D, you may save some overall meshing time but often at the risk of reduced quality due to the inability to easily address meshing issues.

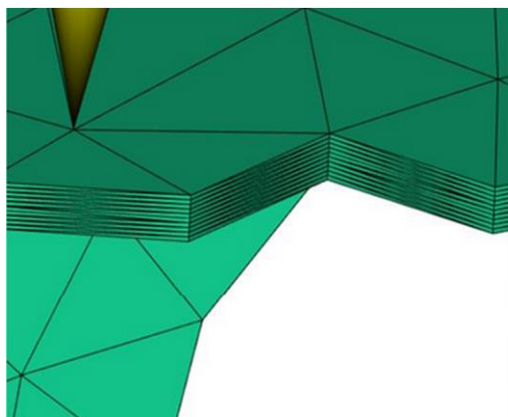
Refinement Guidelines

Proper refinement is required for accurate analyses. It will affect the accuracy of every component of simulation, either directly or indirectly. It also will have a substantial impact on the number of errors and warnings you may get, the computation time, the hardware requirements as well as level of detail available in many results. This latter area will be vital to ensure you not only have accurate results but can succinctly capture the most relevant times in the cycle for the purposes of your simulation.

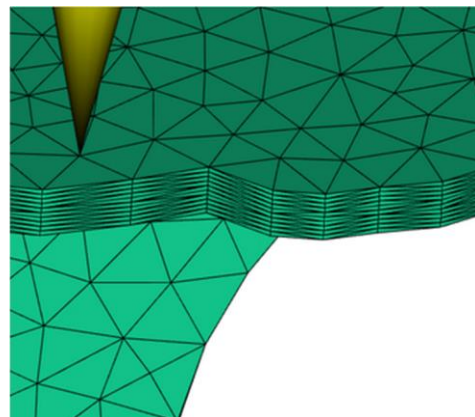
Proper Mesh Refinement

Understanding what Moldflow recommends for proper mesh refinement will help in the creation of suitable meshes that can help predict more reliable results. It is also important to understand why Moldflow has such recommendations.

Functional



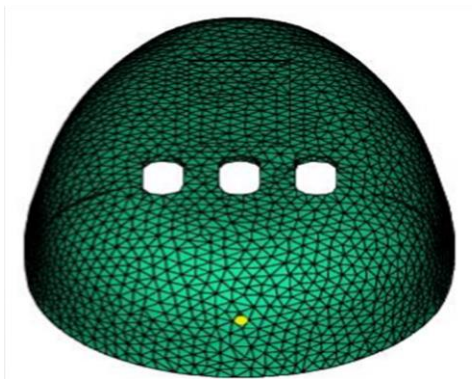
Preferred



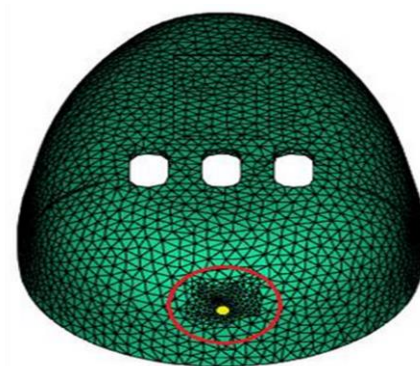
The picture above shows the impact of coarse mesh relative to the number of layers. Naturally as you increase or decrease the number of layers, you may need to adjust the surface mesh to compensate and ensure quality elements. This of course will depend on how refined your surface mesh via the global edge length is to begin with.

This is important as it helps to capture flow through the layers of the part as well as on the surface of the part. Having an overall refined mesh through the entire thickness of the part and on the surface will help provide more desirable results.

Functional



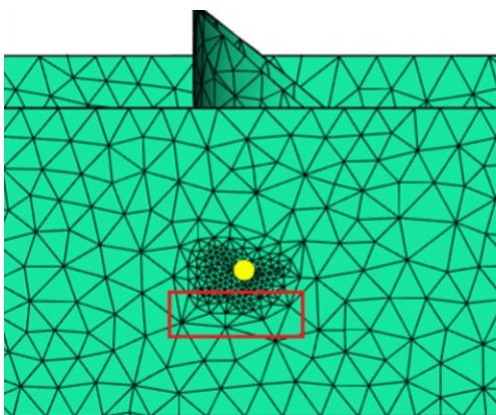
Preferred



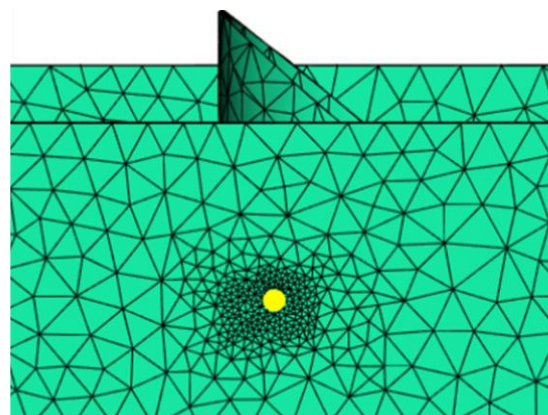
The picture above shows gate refinement. The functional picture shows the injection location to be direct on the part without gate refinement. The preferred method is to have gate refinement around the gating location for 3D analyses. This is not true for Dual Domain.

This is important because it helps to capture the shear heating through the gating location that can help to better predict the flow through the gating location. This is important when reviewing results such as air traps, hesitation, or race tracking because it captures the thermal characteristics through the gate region.

Functional



Preferred



The picture above shows the importance of mesh size transition. The functional mesh shows gate refinement however it also is displaying larger elements beside smaller elements. The preferred mesh shows the importance of having the mesh around the refined mesh to be transitioned into the larger elements.

This is important because it will display more of a transitioning result whereas having larger elements alongside smaller elements can give undesirable and often skewed results.

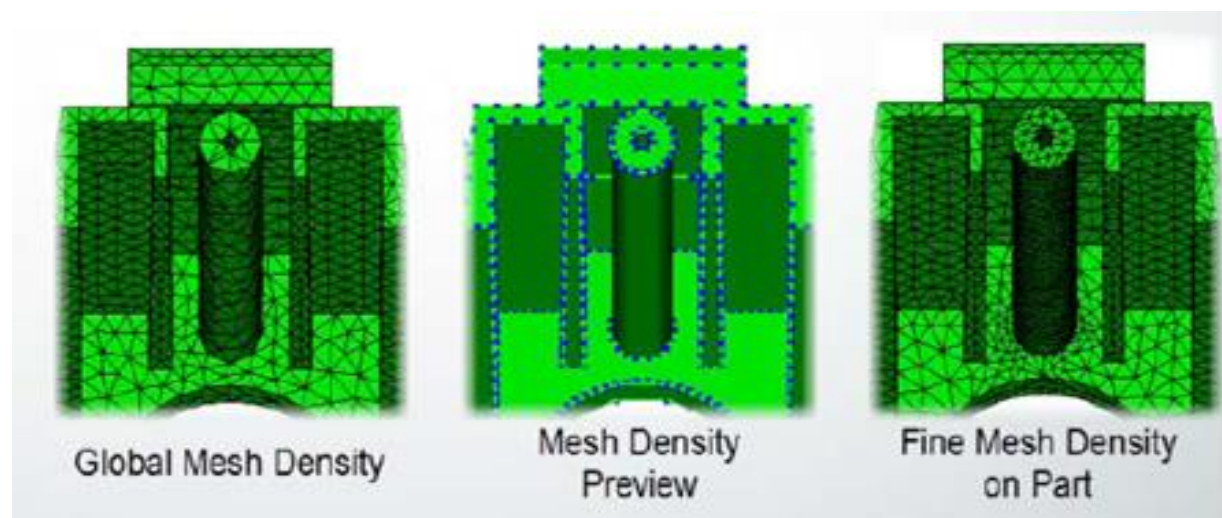
Feature Definition

Another major component when looking at how impactful proper definition is, has to do with how accurately different features and components are represented in the software. This includes features that exist on the part that are necessary for proper function (structural, cosmetic, end use) as well as additional mold components that can influence results such as inserts, cooling layouts, and feed systems. The compelling takeaway is that each of these features is vital with respect to the impact they can have on your analysis, and as such representing them correctly or incorrectly can have a substantial influence on the level of accuracy you can expect.

Design Features

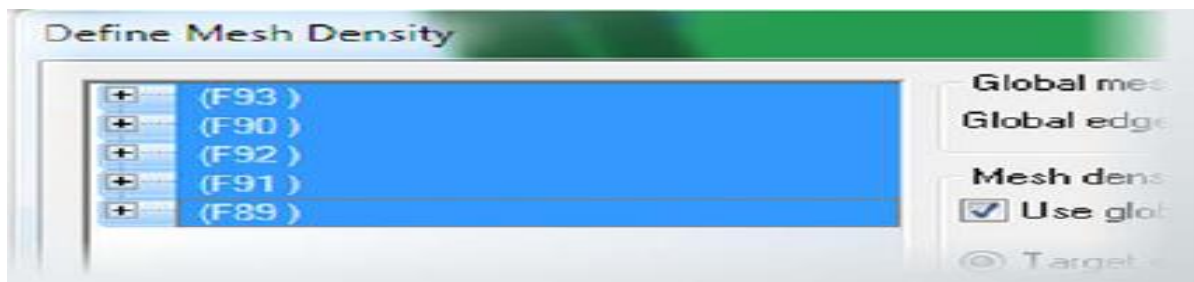
The most direct type of feature would be a design feature. This is representative of something that exists as part of the part that serves some purpose for end use. These could be ribs, bosses, grills or any other uniquely defined geometry. The reason we include such geometry under features is because there is a more detailed approach to meshing them than just your standard part geometry.

Much of this has to do with balance that needs to exist with respect to element refinement, and avoiding having too coarse a mesh as well as having too fine a mesh. These sorts of features often require increased refinement to properly capture, as evident in the cylindrical geometry with a through hole in the image below.

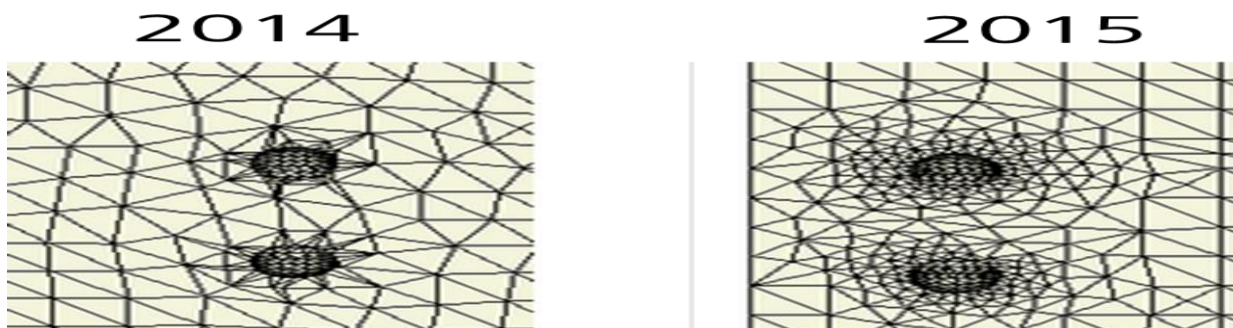


When a global mesh density is used for the entire part and that density is sufficient for representing most of the part geometry, while also ensuring a reasonable element count for the rest of the part. The problem with the global mesh density is that the boss feature is poorly defined, as both the through hole, and the feature itself is “squared” in geometry rather than round. But if you were to apply an increased refinement to the whole part, to ensure the design feature was sufficiently represented you be swinging the pendulum of refinement to the other side. This would create an accurate representation, but inflate the mesh count to an unreasonable number for running analyses in an efficient manner.

The recommended practice for representing and meshing such features is twofold. First the general practice is to adhere to the same recommendations of parts as a whole. That is, when possible always use native CAD representations over neutral representations. And if you must use a neutral file, ensure that you improve the quality as seen by Moldflow through CAD Doctor. The second part though, has to do with taking advantage of the local mesh refinement tool. This is a vastly under used tool by many users, and it can allow the best of both worlds. As highlighted above, it allows you to use different mesh densities in different regions, allowing the user to have sufficient refinement where necessary and still keeping element counts reasonable.



Mesh density also will allow easy control when it comes to selection of specific faces and even edges you want to modify the refinement for. This gives the user total control for both the density of features, as well as blending higher refinement into larger refinement. And a recent improvement in our meshing technology takes that latter point even further, by improving that transition from more to less refined areas.

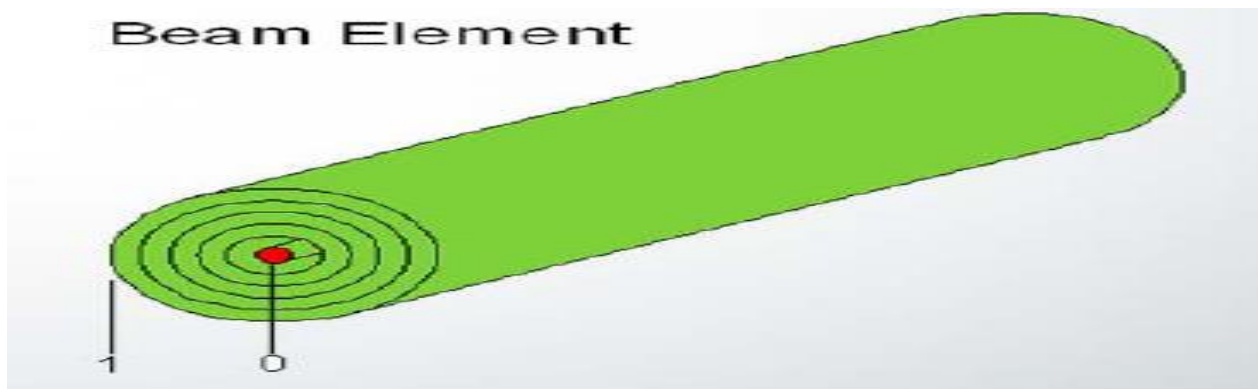


Feed Systems

When it comes to melt delivery to the cavity, properly capturing how the material behaves and the influence it has is another important process to capture. The impact can be both direct (shear heating, viscosity influences) and indirect (maximum pressure impact on pack profile definition). In any case, understanding how to represent your feed system with mesh is an important tool to ensure analyses accuracy.

When it comes to meshing feed systems there are two distinct ways to do so, and a third option that helps mitigate one of the limitations of beam elements. The two primary options are beam elements and 3D tetrahedral elements. We do not recommend utilizing Midplane or Dual Domain triangle elements to represent a feed system in any capacity. The reasons are numerous, but most specifically for two reasons. First they both struggle with representing certain geometry and most often that geometry is fairly standard for feed systems both in shape and size. Second has to do with connection to the part, whereas it often is difficult to build a suitable model for warp when removing the feed system for constraint.

Let's start with beam elements, as they are the primary means of representing feed systems and they have no real restrictions with respect to part mesh. They can be used for all 3 mesh types, and do so at a high level of accuracy. They also have a much higher level of refinement than can be reasonably expected of tetrahedral feed systems, with 24 layers through the thickness.

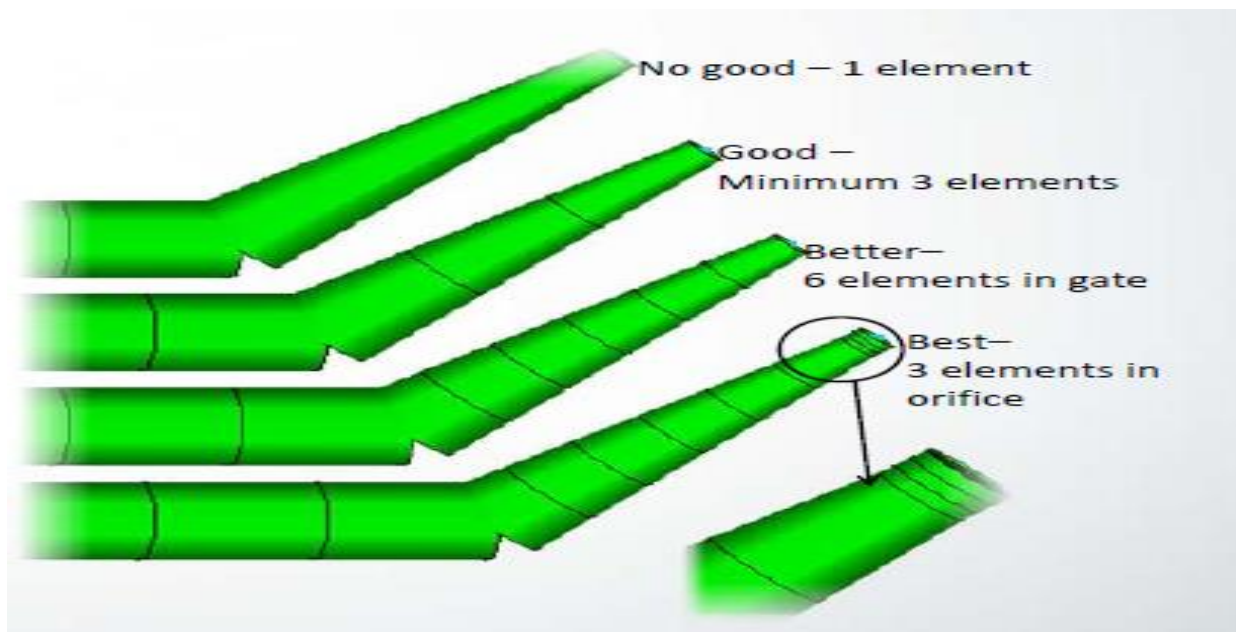


It's worth noting the primary restrictions of beam elements as well, of which there are two. The first is single node connection. For certain type of gates, such as a pin gate, this isn't a major issue. And with respect to the solvers, this also has little impact as well as the beam still captures the appropriate shape and dimension. But as most results are transferred between nodes, having a single node connected to the part can lead to poor representation of flow into a larger surface area, such as those seen in edge gates, fan gates, etc. To address these concerns we have the ability to use connector elements, which are data transference curves. They simply transfer data from one node to another, and thus allow you to transfer data from the single gate node to multiple nodes on the part simultaneously. This can more accurately represent connection to larger surface areas. Connector elements aren't available in 3D.

The other major limitation, and most often the one that drives users to full 3D feed systems is because beam elements are what we refer to as 1D elements. This means we only calculate flow in one direction. The problems this can create primarily have to do with transitions within the feed system where the melt clearly doesn't move in a single direction, and the impact that can have on additional calculations. The largest of which is the impact shear induced imbalances have on the thermal and viscoelastic behavior of the melt. Beam elements simply calculate shear as it exists in one direction, and as there is only 2 nodes for each beam section, transition that data to the next node. If a transition happens to exist, it simply changes the direction of flow, but doesn't calculate how the material would actually behave in those situations.

Because of how one directional flow calculations impact important variables, it can greatly diminish the accuracy of pressure calculations, thermal calculations (magnitude and distribution) and fill pattern.

When it comes to modeling beam elements, best practices are pretty straight forward. First geometric representation is easy and manageable through the standard beam creation process. Here you can define both the shape and size of the beams, and allow the distances between nodes or curve length to determine layout. Outside of that, the primary thing we want to focus on here is refinement. There are two rules of thumb, one having to do with aspect ratio and the other with effectively capturing behavior in a given section of the feed system. The latter of which, is the more important as aspect ratio (often beam length to diameter) is most important with respect to thermal calculations but largely isn't as influential as it was in the past. So when possible, aim for a 2.5:1 ratio. When talking about refinement though, the goal is no less than 3 distinct beams per unique section of runner.

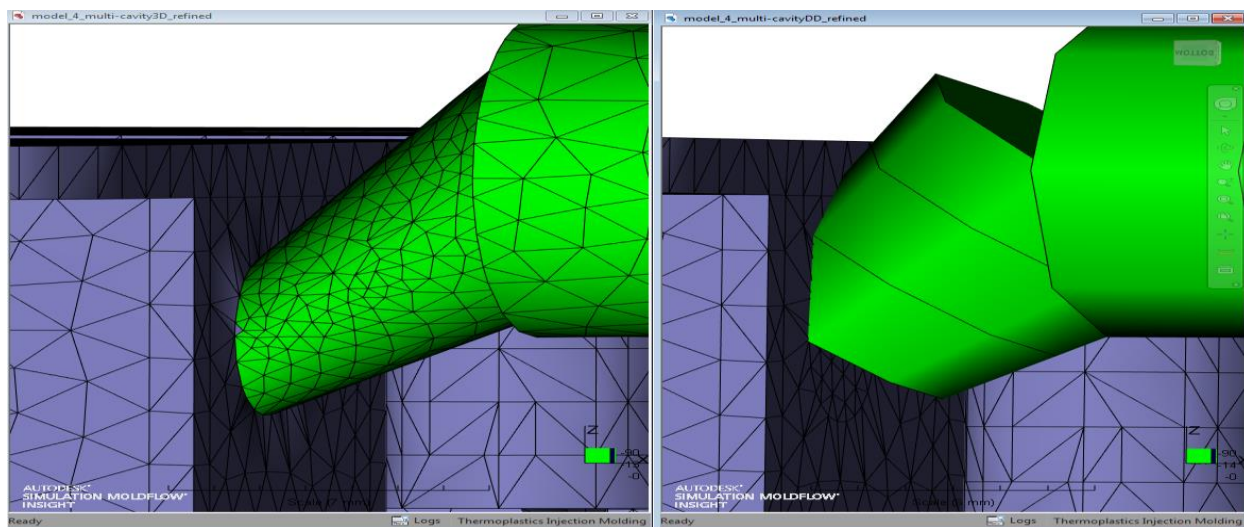


This rule is most often referenced with respect to gates, but is also true of sections of the runners and sprue as well. The reason being has to do with the 2 node nature of a beam element. If you have only a single beam to define a section, you're simply calculating what happens in that section from the start to the end. In theory that sounds fine, but in practice it actually can diminish accuracy with respect to picking up behavior over the length of such a section. This is amplified in gates, as the sizing across the gate often changes due to tapers and such, increasing the likelihood of not capturing all the behavior if sufficient refinement is not used.

The other option of course is 3D tetrahedral elements. There are more restrictions with respect to this feed system representation, most obvious of course being that you can only do so with 3D part meshes. This option is not available for Midplane or Dual Domain parts. The other primary issue is an increase in element count. Considering the impact that can have on both hardware requirements as well as time to analyze, it's worth noting how important this can be.

The benefits though are quite impactful, especially with respect to accuracy and influence typical of a feed system. The most substantial influence is via the capturing of the influences of shear heating and the resulting impact that has on viscosity. This is particularly true with respect to more complex feed systems. Due to the limitations of beam elements and use of 1 directional flow, the proper recognition of shear induced imbalances in a feed system completely removes this phenomenon that we know exists.

The other primary advantage is a more complete and comprehensive representation of the feed system in question. Simply put, the ability to represent feed systems in a true way is more complete when tetrahedrals are used rather than beams, especially with respect to non-standard shapes, geometries, and layouts. This is most evident when looking at unique gate surface contact, such as a tunnel gate or cashew gate providing a different contact surface than beam elements.



When it comes to best practices, the process is actually very similar to those for parts. These requirements being the need for at least 6 layers through the thickness (although more is ideal), the need for a 3D part mesh, and the need for a surface mesh that provides reasonable aspect ratios with respect to the internal mesh. There are benefits to increase the refinement as you get closer to the gate, to ensure you capture behavior in areas of changing geometry or contact. You can use the surface refinement tool to assist here by setting different mesh inputs in those regions.

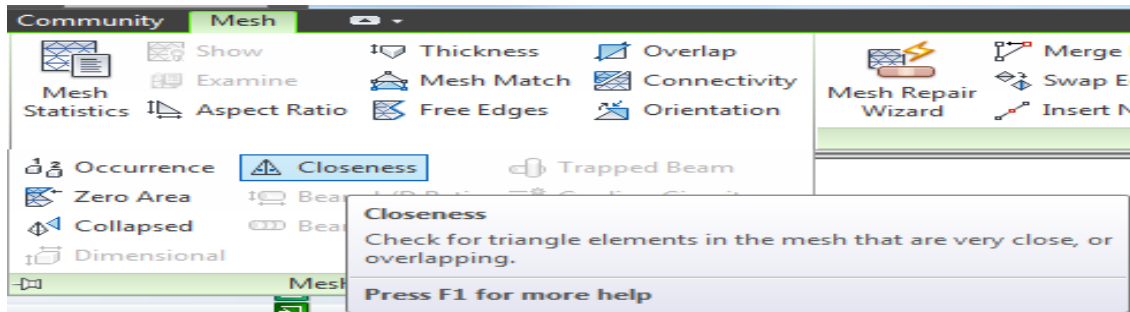
Cooling Systems

In a similar vein, cooling systems have the same two options with respect to representation. That is you can represent them as beam elements, or 3D tetrahedral elements. But the complexity of cooling analyses as a whole require a different approach to both, and have different considerations that need to be made with respect to both the cooling channels as well as with respect to the part mesh.

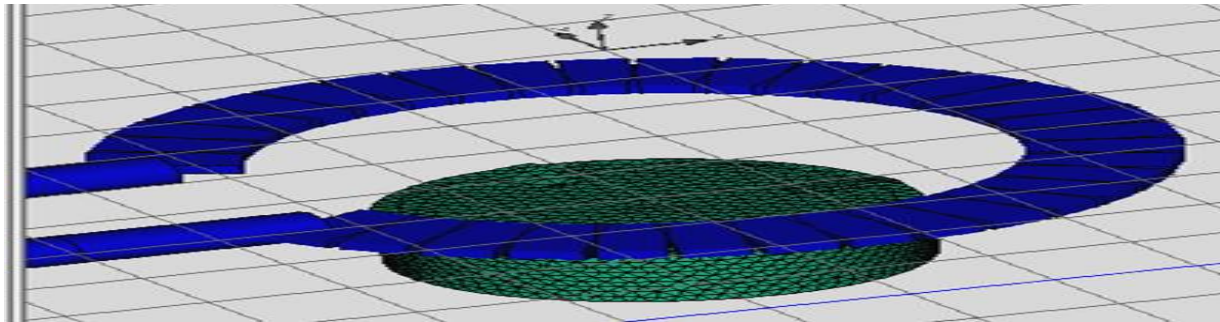
First, it's worth noting the part mesh considerations to recognize with respect to cool. This is especially true for the legacy cool solver, Boundary Element Modeling (BEM) cool and beam cooling for Finite Element Modeling (FEM) cool. The reason has to do with beam element cooling requiring convergence of temperature during the solver calculation. One thing that can drastically impact this negatively is the part mesh and how it is configured. Specifically the impact that the centroids of 2 or more triangles can have when they are too close. This can cause the temperature to be high in that local area, negatively influencing the ability to converge.



There are two practices you can employ to ensure your part mesh is of sufficient setup to address this issue. First and most important is to use the Closeness Diagnostic. This is a tool to identify elements centroids that are too close to one another (determined by calculating elements that are closer than one tenth of either element's average edge length) and fix them manually if necessary. Second is to ensure your surface mesh (Midplane and Dual Domain final mesh, 3D external mesh) has a max aspect ratio no greater than 20:1. If done prior to meshing to 3D, while not guaranteed, it can help to minimize the 3D aspect ratio to a more reasonable number (less than 100:1 in most cases).



After the part has been meshed appropriately to handle cooling, we need to model the cooling lines themselves. As noted, there are two options for modeling; beam elements and 3D tetrahedral elements. When it comes to beam elements, the requirements are a bit more stringent than that for feed systems, specifically when running a BEM cooling analysis. This has to do with the length to diameter ratio, a number you want to keep at 2.5:1 or larger. The reason has to do with area of effect. The smaller that ratio, the more likely you'll get too much overlap element to element, thus negatively influencing convergence.

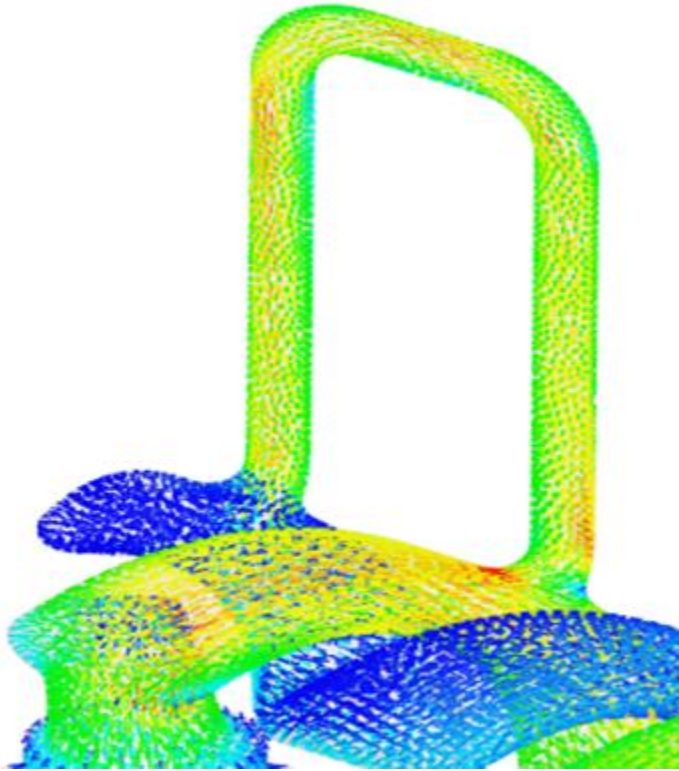


This of course makes complex cooling channels difficult to model when running BEM cool, because curved layouts simply can't achieve both a length to diameter ratio of 2.5:1 and capture the layout correctly. This requirement is not as impactful in FEM cool when beams are used, as convergence has much less of an impact in FEM cool. Generally speaking this ratio is still ideal in those situations, but not as impactful.

When meshing the cooling channels, there is little else to really mention. The layout can be manually created or imported via IGS curves, but the meshing itself simply requires an edge length for beam element definition.

The other option has to do with new technology added during the 2014 release of Simulation Moldflow. It is the mesh of cooling channels using 3D elements. The main thing to note here is the use of the Simulation CFD Design Study Environment as the mesher. We refer to this as conformal cooling, as it allows users to do exactly that. They can set up much more complex cooling layouts without concerning themselves with issues caused by beam elements. This also bypasses another limitation of beam elements, that of 1 directional flow. So again we see a

similar benefit as that in feed systems, where coolant behavior is more accurately captured as it moves around transitions features in the cooling layout.



To access these capabilities, there are several steps that need to be followed. First, you need to specifically designate the analysis as Cool (FEM) so that the options to represent the cooling as 3D can be accessed. Secondly, the cooling channels need to be added via the “*Add Cooling Channel(s) 3D*” option. This allows selection of the CAD representation of the cooling layout, so that it can then be meshed.

Before you can mesh your channels, you next need to define both the inlet(s) and outlet(s) of each unique channel. This is a requirement for the channel mesh option to become available. Finally you need to mesh the actual channels with user defined edge lengths and refinement. For this, you currently need access to a Simulation CFD

Design Environment license, as that is the technology used. The option to mesh the channel is available through the mesh Ribbon in the user interface, as it is a separate function from the standard part mesh.

The mesh refinement for the cooling channels is similar to that of a part on several fronts. You don't want to over mesh or under mesh, as that can lead to extended analyses times or poor representation. But you do want enough refinement to properly capture the geometry and ensure quality tetrahedral elements to analyze the coolant.

Inserts and Assemblies

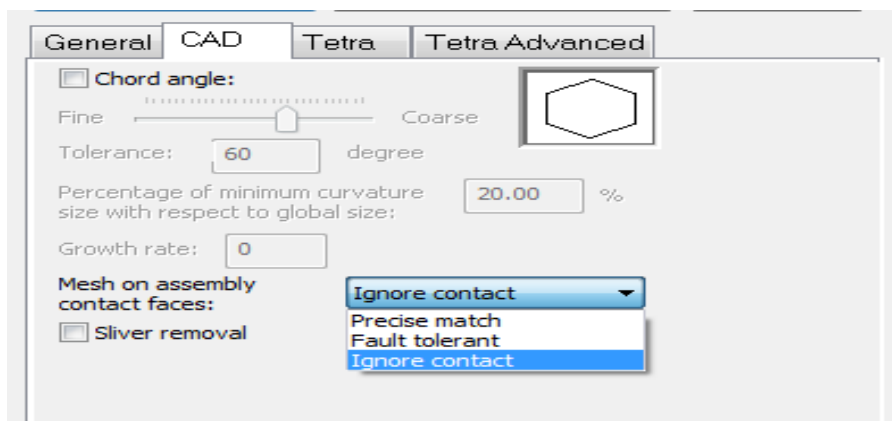
Over the past few years, arguably the most common non-part feature represented in Moldflow has been that of assembly files, specifically for representing inserts. As inserts represent a variety of different types of components, it's important to understand the best practices and requirements for representing them in an effective manner.

The first thing to look at here is the different ways in which Moldflow handles different components. Because there are a variety of different options, the way they behave, the types of analyses you can run, and the mesh types that can be used are necessary to understand if you plan on incorporating them into any of your analyses.

How we treat inserts?					
		Software package:			Comments
		Midplane	DualDomain	3D	
FLOW	Part	Yes	Yes	Yes	Mesh type can be mixed between Part & Insert.
					However, mesh type can not be mixed from part-to-part or within part itself
	Part insert	Yes	Yes	Yes	
	Mold insert	No	No	Yes	Mold inserts do not affect flow directly, only through a prior cool analysis
	In-Mold Lable	No	No	YES	Coupled3D can deal with in mold labels
					Midplane/Fusion : In-Mold Labels do not affect Flow directly, only through a prior cool analysis
COOL	Part	Yes	Yes	Yes	
	Part insert	Yes	No	Yes	
	Mold insert	Yes	Yes	Yes	
	In-Mold Lable	Yes	Yes	Yes	
WARP	Part	Yes	Yes	Yes	
	Part insert	No	No	Yes	A part insert does not constrain or deflect with the part in Midplane or DD
	Mold insert	No	No	No	A mold insert does not affect warp directly, only through Cool
	In-Mold Lable	No	No	No	In-mold Labels do not affect warp directly, only through a prior cool analysis

The above highlights the restrictions and capabilities of different types of inserts. This is the first step in meshing your insert, to ensure you use the proper insert type relative to the part mesh you have as well as the type of analysis you plan on running.

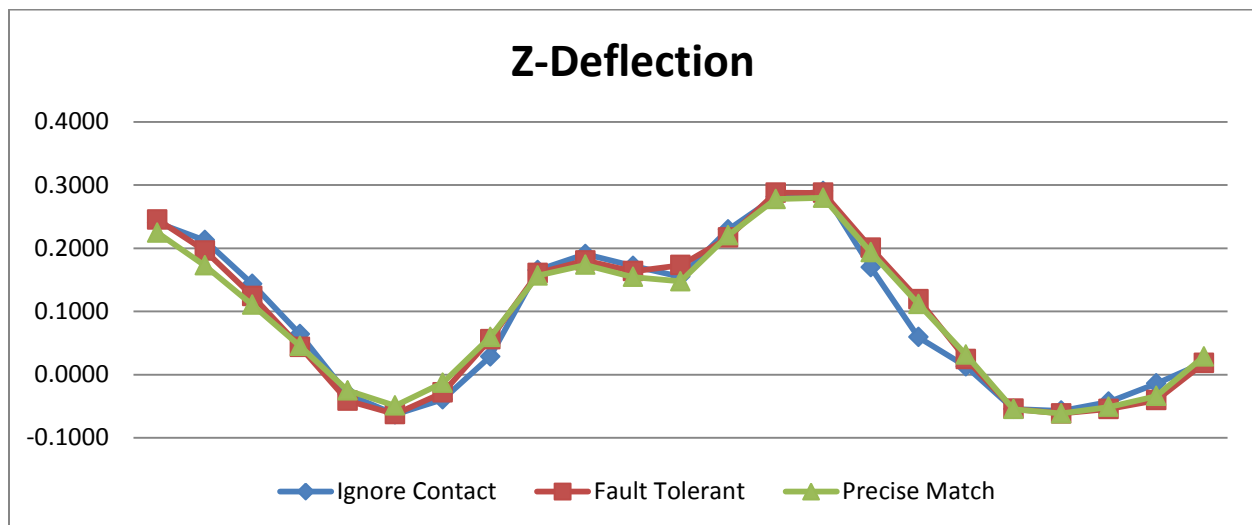
With respect to best practices, there are a few things to note. First when importing or generating the geometry, the ideal practice is the same as that of a part. That practice being the use of native CAD geometry always being ideal. This can be just as impactful from a geometry representation and mesh ease standpoint, but there are also benefits when using an assembly file. As native CAD files are imported via Moldflow Design Link, individual components are separated and properly fitted together to ensure accurate interfacing contact. That interfacing contact can be controlled during meshing per your own required specification via mesh assembly on contact interface.



These options determine the tolerance applied when allowing a mesh to complete successfully. If the specified tolerances are not achieved, the mesh will fail.

- **Precise match:** Requires no overlap between bodies in any capacity and an attempt to precisely align nodes on the interfacing surfaces. They must either contact perfectly, or have gaps. No actual interface with any overlap can exist or the mesh will fail.
- **Fault tolerant:** There is a bit more leeway, where small variations are compensated during meshing in an attempt to align the nodes and surfaces when possible. If there are issues with ensuring such contact and the compensations are insufficient to provide the match necessary, this process will also fail.
- **Ignore contact:** The other extreme compared to precise match, allows for mesh to complete regardless of errors in the CAD model and issues with contact. This may require manual modification of the mesh if the impacting CAD issues are substantial.

In the grand scheme of things, these options actually have very little influence directly on the accuracy of results. Because of the extremely tight tolerances of quality for both precise match and fault tolerant, it can often be extremely difficult to get a completed mesh even if the issues that exist are negligible in terms of impact. The best process will always be to use precise match if possible, because it guarantees that the mesh quality of interfacing components is high. But in many cases this may not be feasible, so don't spend too much time trying to get a precise mesh or even a fault tolerant mesh if they are causing you issues. Recognize that the overall impact on the results will be negligible unless there are drastic issues with the model. In that case, be prepared to manually troubleshoot the mesh as you would any normal part mesh.



The process for meshing the individual components then is the same as a standard part. With Dual Domain, once the geometry or assembly has been imported, you can simply mesh per your necessary refinement and troubleshoot any issues. For 3D, you can mesh straight to 3D, or more ideally like with a part mesh, go to Dual Domain first. This of course allows you to

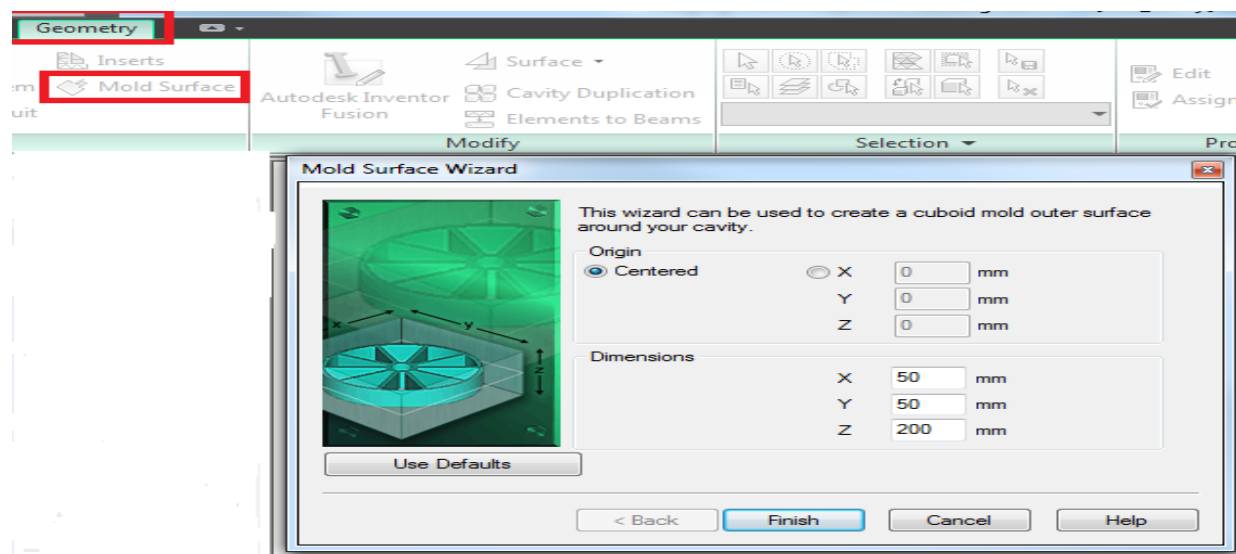
manually make improvements and address any potential mesh related issues. Once you've tightened up the quality of the Dual Domain representation, you can then mesh the part to 3D with the required internal configuration.

When it comes to the element definition for inserts and non-part assembly components, standard mesh settings are sufficient. We recommend at least 2 elements across a thickness, and at least 3 elements to define a feature. In terms of 3D meshed components, here we can actually reduce overall element count by using only 4 layers through the thickness, provided you don't need more to ensure quality aspect ratios depending on surface refinement. The reason being that parts require a minimum of 6 layers (an ideal of 8+) to properly capture plastic flow and how it varies across the thickness. With a non-part component, this is not relevant as plastic will not be flowing through it. That said if you have a highly refined surface mesh for some reason, you'll need to adjust your internal refinement to ensure you don't get poor aspect ratios and may require more than 4 layers to do so. At the same time, that situation may also be a good time to reevaluate the surface refinement on the insert, as it may be too high.

Mold Block

The final component to cover is the actual mold itself. Depending on the type of analysis being run, a mold block doesn't even need to be created. Moldflow will assign a default size for relevant thermal calculations in lieu of a mold block being actually modeled. Outside of that, there are two primary means with which the mold can be represented. Each is essentially representative of the two types of cooling analysis available. One is the boundary representation of the mold, the other a 3D tetrahedral representation of the mold (for FEM cool).

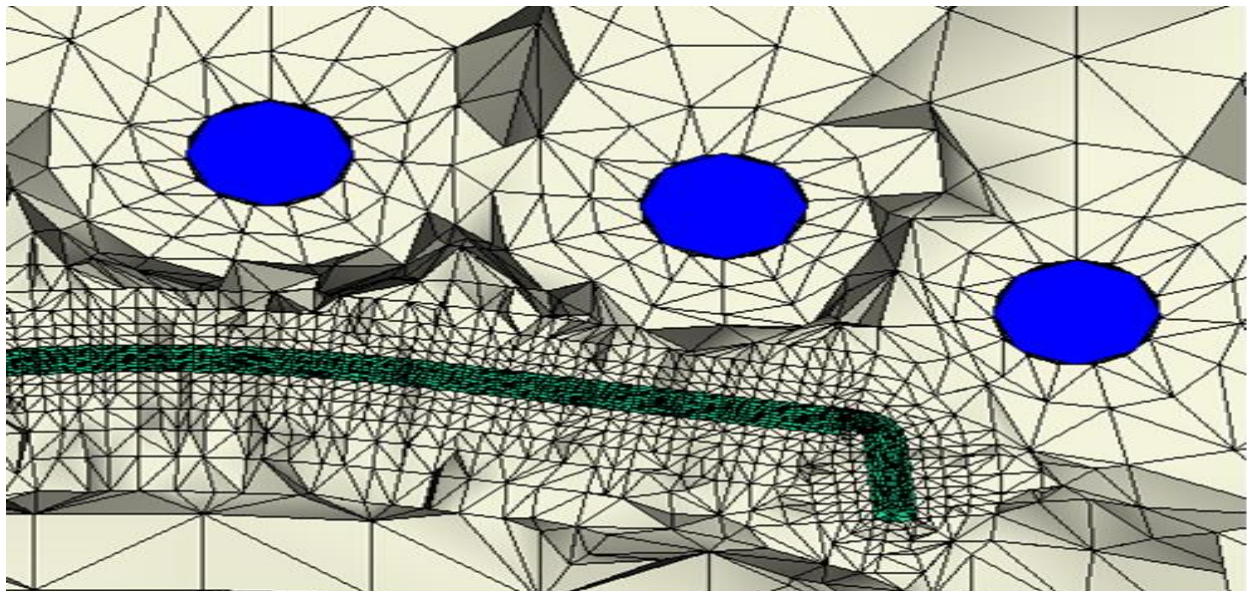
When it comes to the boundary representation of the mold, the process has been around for some time and is quite easy to follow. Moldflow provides a mold surface tool, in which you can specify both the dimensions of the mold block, as well as where it is centered. This generates your desired mold boundary.



The most important thing to note with the mold boundary is that other components in the model that intersect with the meshed surface can cause issues with convergence. It is recommended that holes be cut in the boundary surface where such intersections would exist, such as cooling channels. This may look odd, depending on what the surface refinement is, but necessary to ensure proper use. It is also worth noting that these holes have very little impact on the results, as the primary purpose of the boundary is to determine the volume and thickness of mold material. So having small holes does not really compromise the software's ability to make that calculation.

The second option is to model actual 3D tetrahedral elements for representing the mold block, a requirement for running FEM cooling analyses. The most direct means of doing this is to import the actual CAD representation of the mold. This is effective because a CAD representation of the mold will have open spaces for non-mold block components, such as inserts and feed systems, thus not fill them in with a 3D mesh. This is important because you'll be representing these entities via separate means and it will avoid meshing related issues.

The process for meshing that CAD representation is to utilize the 3D mold meshing tool in Simulation Moldflow. It will allow the specification of a global edge length (surface refinement for internal contact interfacing) as well as the number of layers and the gradation of the mesh from inside to outside. As with most direct CAD meshing, the process itself is quite straight forward.



If a CAD representation of the mold block is not available, you can utilize tools within Moldflow to create and mesh your mold. The first step is to stitch contact interfaces between cavity, inserts and other components. There is a tool to do such on the mesh ribbon, Stitch Contact Interfaces. Next you'll need to create the mold boundary, much like you would if that's all you were creating (as highlighted in the previous section). After you create that boundary and before you begin the mold meshing process you'll want to ensure that all internal components are

visible (feed systems, cooling, inserts, etc.). The mold meshing tool has collision detection and if any component is not visible, it will mesh 3D elements in its place.

Once everything is visible, to create the mold block mesh you'll need to mesh the mold twice. First you'll define the internal and external surface mesh, and then define the internal tetrahedral mesh. The surface mesh will represent the external mold wall and the internal contact surface with other components (part, cooling lines, feed system, etc.). Because of this, the external mesh should have a lower refinement than the internal surface mesh. After that meshing process completes, you'll use the same mesh tool to mesh again. Here you'll note the options are now different, indicating controls for the internal 3D mesh. Here the number of layers can be specified, as well as the gradation ratio from the surface to interior regions. The default is often sufficient as it will provide smaller, more refined elements closer to internal components. That said, you can decrease the ratio if you'd like elements to be consistent throughout the molds thickness, although it is not necessary.

The mold block, especially when made from 3D tetrahedrals, is often the last non-part component that a user will represent. As noted above, one major reason is that other components need to exist so that the tools can properly recognize them and account for them when filling in the 3D mesh. It should also be noted again that 3D mold representation is for FEM cooling analyses.

Learn How to Incorporate Other Tools to Improve Quality

Overview

One of the most important things to consider when setting up your mesh in Simulation Moldflow is that Moldflow is primarily simulation software, not a modeling tool. As such we have capabilities to take advantage of interoperability with a variety of other tools whose sole focus may be on modeling, or meshing, or even model repair. Using these other programs can be an

extremely important step in ensuring you have not only a quality part representation, but also a quality mesh to show that part. This section will primarily detail what these different programs are, what they offer to users, and how Moldflow operates with them. It will not go into detail as to how to use them, simply because that could be a separate document and presentation in and of itself.

Also worth noting, many of these programs are not Autodesk programs and as such our discussion on them will be less detailed, and will not cover capabilities outside interoperability.

Autodesk Tools

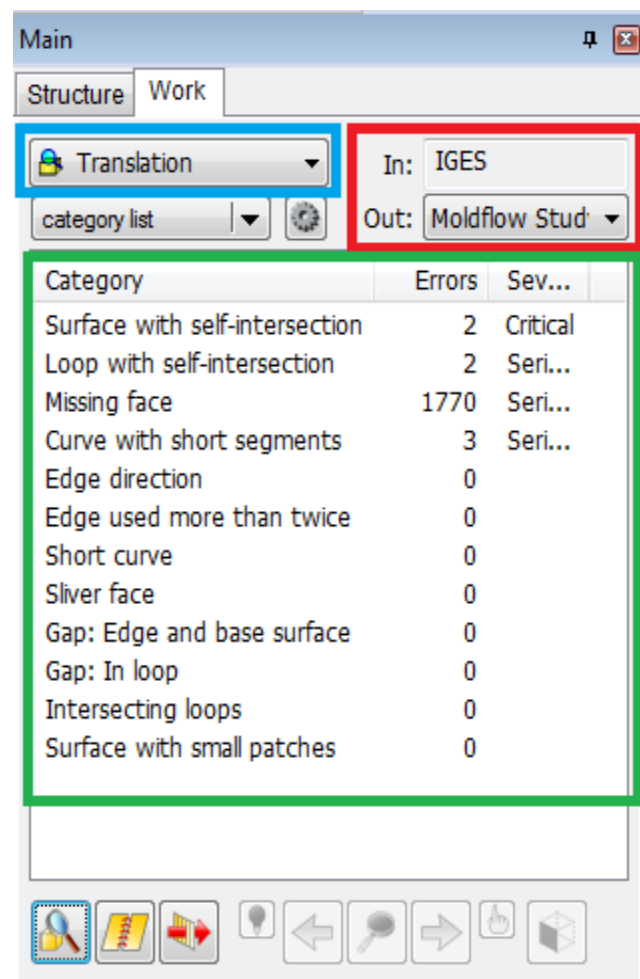
The first few programs to look at are those provided by Autodesk. Naturally these programs offer a bit more in terms of direct interoperability and capabilities with respect to improving your models, and ultimately your mesh. That's not to discount non-Autodesk products, but naturally being able to control the development of all programs involved helps with usability with respect to Simulation Moldflow.

CADDoctor for Autodesk Simulation

Arguably the best tool for improving quality is CADDoctor for Autodesk Simulation. The reason being, CADDoctor has the ability to both simplify the geometry (such as removing fillets, rounds, features, etc.) as well as improving the overall quality of the file by addressing problematic issues. And because of the interoperability built in, you can export a Moldflow study file directly, making for easy import into Simulation Moldflow.

The process for using CADDoctor is quite simple. You will need to import your CAD file (Moldflow Design Link can be used for non-IGS files) and check it. This will provide all relevant details for your file in which you can then choose how best to address such issues. You'll get an output similar to the following image.

There are two things to note with this particular image. The first is the recognition of the sort of function you'll be using (blue highlight). Here we'll be using translation, which is the means with which you can address errors or problems with the model.



The other option, accessed via the drop down, is simplification which allows you to modify the geometry itself by removing or simplifying features.

Next you'll note (red highlight) where the type of file imported is listed, as well as the format you'll be exporting. Here you can change between a Moldflow study file (*.sdy) and a Moldflow model file (*.udm).

Finally (green highlight) you'll note the listing of issues. Depending on the function, this will either be a list of problematic errors, or a list of features (bosses, ribs, rounds, etc.). For the problematic errors, as shown here, you have several options. The first option is to heal, which will address many of the issues outright, and is also necessary to export the file properly to be used in Simulation Moldflow. After healing, there may be several issues that still exist. These can be addressed manually with a variety of tools, depending on the type of issue. Once you have fixed the model to your satisfaction, you will need to heal again, and then your part will be ready for export.

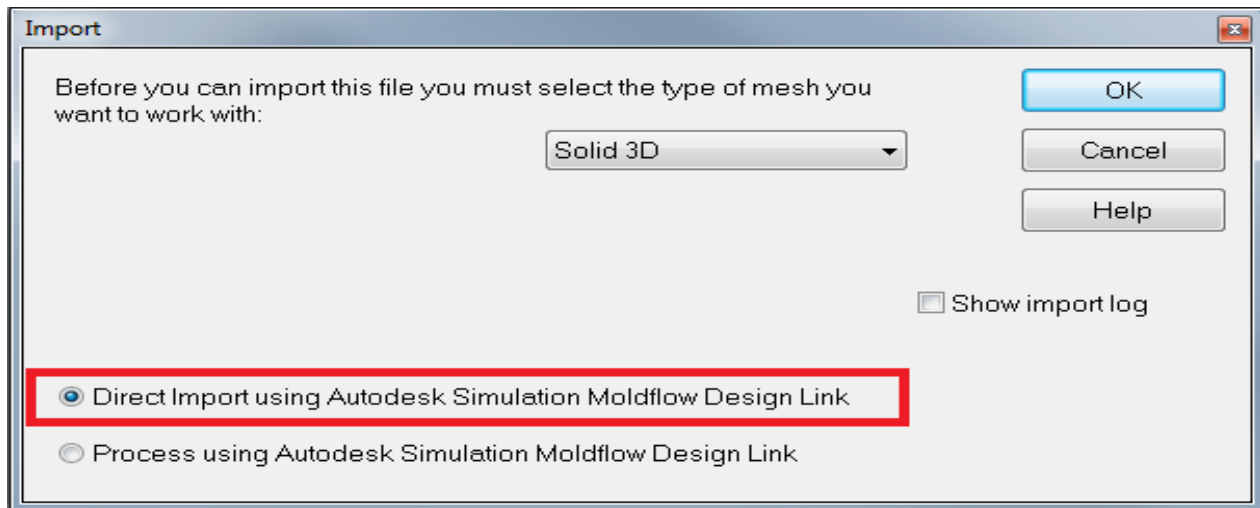
It's worth noting again, that CADDDoctor does not provide a mesh though. It only improves the quality of the CAD file that will be used in Moldflow for meshing. But as noted earlier in the quality section, this is quite important both to ensure proper geometric representation as well as improving time to mesh and analyze.

Simulation Moldflow Design Link

One of the most important steps in ensuring a high quality part is represented is to have access to the most accurate CAD representation of a part. We provide the ability to do just that through Simulation Moldflow Design Link. Previously Design Link was a paid component that required a separate installation and licensing, and also had a distributed set of libraries specific to the version you purchased. Starting with the 2014 release, Design Link is automatically installed with Synergy and no longer requires a separate license.

This is important to note because Design Link is one of the most important tools for ensuring high quality part representation, as well as the ability to import and separate multi-body assemblies. In the past, we only generated NURBS surfaces with Design Link. This only captured the geometry of the surfaces, but not the topology which could lead to issues with quality representation.

In the more recent versions of Design Link we now convert the file to the Autodesk Shape Manager (ASM) kernel as well as generate the NURBS data. This provides the most accurate and realistic representation of the part possible and is at this time by representing both the geometry and topology of the file. We've also removed the separate installation and licensing requirements for Design Link, making it essentially free and easy to use. Simply importing files that aren't STL or IGS files provide you this option, via the direct import method offered on import.



The direct import is ideal in almost all situations. The processed import converts the model to only NURBS and thus fails to represent the parts topology accurately. If you do use processed, ensure you disable “mesh” during import and enable surface import. The default setup will often fail if it is unable to mesh the part, and if surface import is disabled you won’t even get a surface to manually mesh.

Inventor Fusion 2013 R1

Although discontinued, Inventor Fusion 2013 R1 is still a useful tool for improving model quality. It functions very much like the simplification portion of CADDuctor in that being a history-less tool; you can easily make modifications to the part geometry. This is especially useful when removing or modifying features that could cause issues with the quality of the mesh in Simulation Moldflow.

There won’t be much to cover here, as opening a CAD file is straight forward and many of the options within Fusion are self-explanatory. A few things to note about Fusion though;

- 1.) The 2013 R1 version is the last version of the desktop Fusion series
 - a. It is still included with the installs for Sim Moldflow 2013, 2014, and 2015
- 2.) There are new versions of Fusion that behave the same way, but they are all cloud based tools
 - a. They require forced updates to ensure proper version
- 3.) Fusion is not a meshing tool. It is a tool used for model translation, such as from one file format to another, as well as for modifying part geometry as necessary.
 - a. It also does not fix problems

Non-Autodesk Tools

The primary non-Autodesk tools are meshing tools. Rather than cover individual meshing programs, we’ll simply list them below, and detail interoperability for all of them because they all behave similarly in that sense. Of course we won’t detail processes for each, as this is not about

using them specifically, but more about using what they provide in Moldflow to improve the quality of the part representation.

Programs

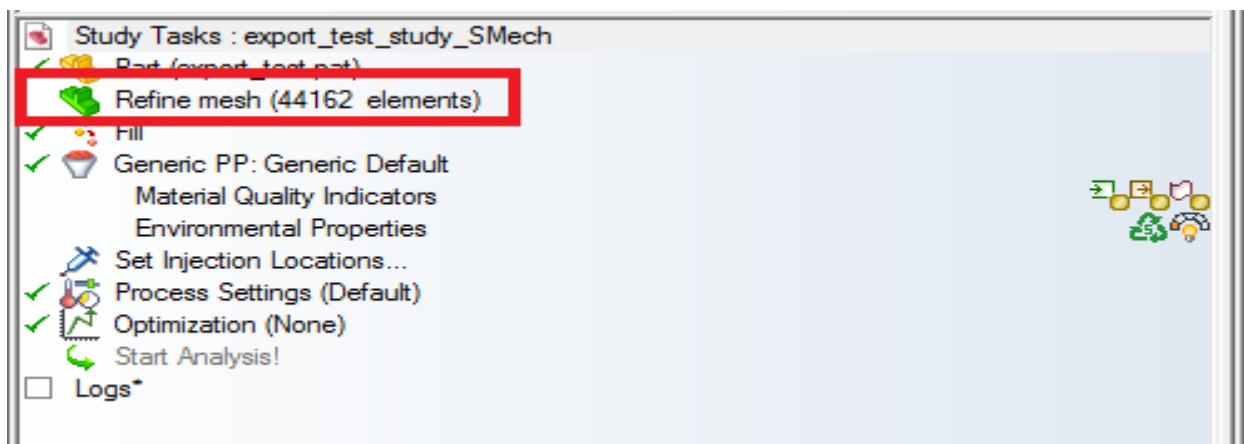
Technically any program can be used, provided it is a meshing tool that can generate some sort of mesh format, such as a Nastran file (*.nas). The most common ones used by Simulation Moldflow users are;

- Altair Hypermesh
- Beta CAE Systems S.A. ANSA

Use

The primary process is to import the file generated into Simulation Moldflow. During this process, Moldflow will automatically “flag” the source of the file. As Simulation Moldflow requires very specific mesh refinement to properly capture plastic injection molding, checking the file source determines whether the part may need remeshing or not. This only really has an influence on 3D meshes though, as both Midplane and Dual Domain can push past this requirement (although it can have negative influences on results depending on thickness assignment and matching). With 3D, the internal mesh is what dictates sufficient flow calculation and many mesh programs don’t provide the appropriate definition necessary.

This flagging precipitates a required remeshing in 3D under all but one circumstance. Therefore in those cases, your external mesh may end up being remeshed so that it can be run. You can minimize the changes by disabling the 3D mesh options to optimize surface mesh, but it is still possible the internal mesh will be modified. This can requirement can be verified when the imported mesh shows defined 3D elements, but in the study panel lacks a check mark to indicate readiness for analysis.



The only situation in which this can be bypassed is when the file imported is a *.udm file. These are Moldflow mesh files, and as such can only be exported from specific programs that have that particular feature. At this time, this option is only available in Altair Hypermesh. Importing a

*.udm file bypasses the flagging process, and thus doesn't require a remesh. Caution here as it is unlikely you will have the most ideal plastic simulation relevant mesh, which the flagging process addresses in the first place.

Best Practices

In general the recommendation here is to approach the use of non-Autodesk tools with caution. They provide a substantial amount of flexibility and in some cases can create ideal scenarios not possible to create within Moldflow. It also allows some significant control and organization.

But Simulation Moldflow requires very specific mesh configuration, especially in 3D, to properly capture plastic flow. A structural mesh, which is what many people are most accustomed to, is not ideal for plastic flow. The flagging mechanism built into Moldflow has a purpose, so we recommend you do remesh after import simply to make modifications to address any issues. The mesh itself is often changed very little, so this is a worthwhile step. If you choose not to do so, be aware the impact this can have on the accuracy of flow calculations.