AUTODESK UNIVERSITY

SIM20611

We’re Having a Kegger! Upfront Simulation for Optimized Design

James Herzing
Autodesk, Inc.

Michael Smell
Autodesk, Inc.

Vaclav Prchlik
Autodesk, Inc.

Learning Objectives

- Optimize your design for better performance
- Reduce product development time
- Have a better understanding of the Autodesk Simulation portfolio
- Leave with a greater appreciation for the science of a cold beer

Description

Implementing Simulation early and often in the design process helps to reduce prototypes, get to market more quickly and optimize your design for better performance. Designs are all around you, some that have used simulation, and some that could benefit from this optimization. In fact, most products require the use of multiple simulation products to achieve their best design.

I know, I know… so where does the kegger come in to play?! It might seem strange, but something as simple as a keg can be used to perfectly demonstrate nearly every Autodesk Simulation product. In this class you will learn a bit of Autodesk Fusion 360, Autodesk CFD, Autodesk Simulation Mechanical, Autodesk Moldflow, Autodesk Moldflow design, and be able to determine which simulation solutions are right for you.

Your AU Expert(s)

James Herzing is the Product Marketing Manager for the Autodesk Simulation portfolio. He has spent 12 years in the field of Finite Element Analysis and Computational Fluid Dynamics, starting his career at Algor Inc., and with the last 8 years spent at Autodesk. He graduated from the Pennsylvania State University with a BS in Mechanical Engineering. James has presented in nearly 20 classes at Autodesk University, being awarded “Top Rated Speaker” multiple times.

Linear Static Stress Analysis

A linear static stress analysis is a great up front simulation that can be run to verify the quality, safety and overall performance of a design. This analysis typically runs in much less time than most other analysis types, but also comes with a few restrictions:

1. The design must deform with small displacements
2. Loading is static and remains constant over the duration of the analysis
3. All parts material properties remain in the linear range when loaded

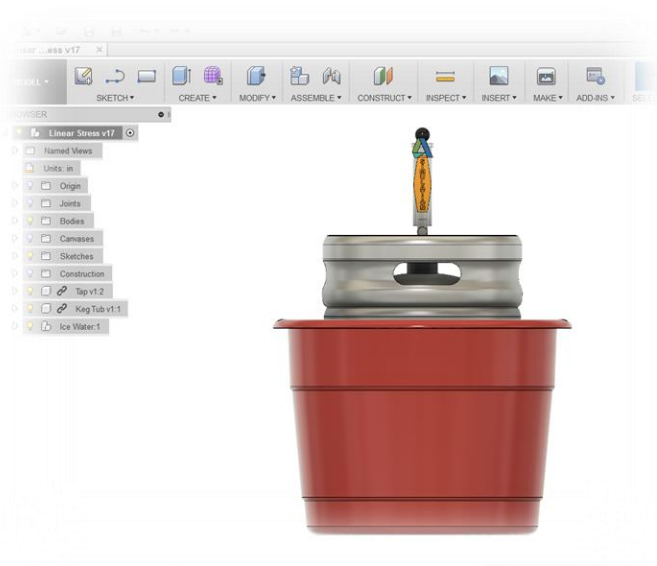
Following these rules, linear static stress can tell you if your design will potentially fail under any number of loading conditions as well as if you should further investigate the design with more advanced simulations.

Autodesk Fusion 360 Workflow

Enter the Simulation Interface

Step 1: Press **Model** → **Simulation**

Why: This will take your Fusion 360 CAD design from the modeling interface to the simulation interface, where you will be able to choose what type of analysis you want to perform.

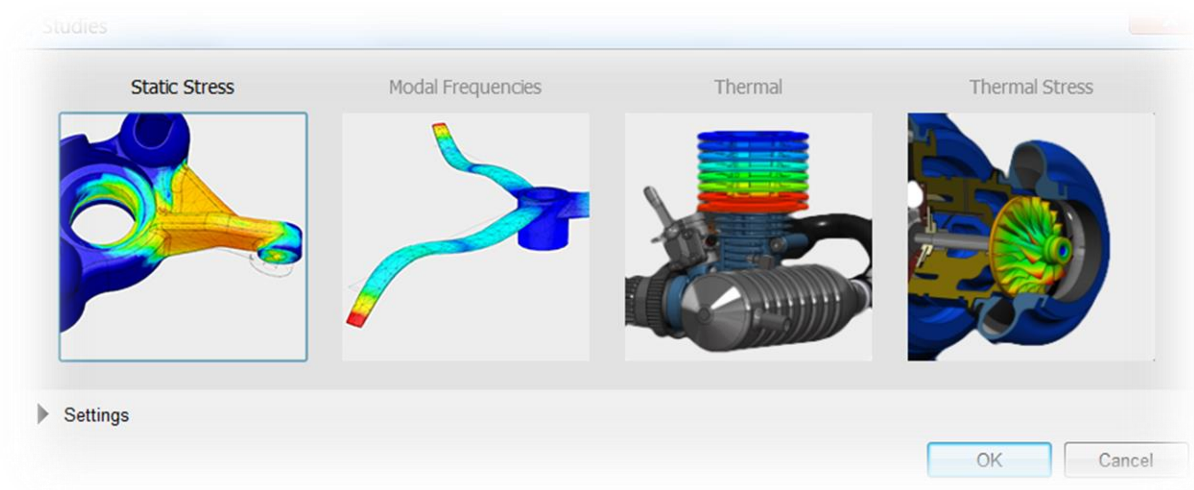


Simplify the Model

Step 2: Press **Study** → **Static Stress** → **OK**

Step 3: In the model components dropdown of the browser, uncheck the box next to the *Ice Water* and *Keg Tub* components, as well as *Beer* under bodies.

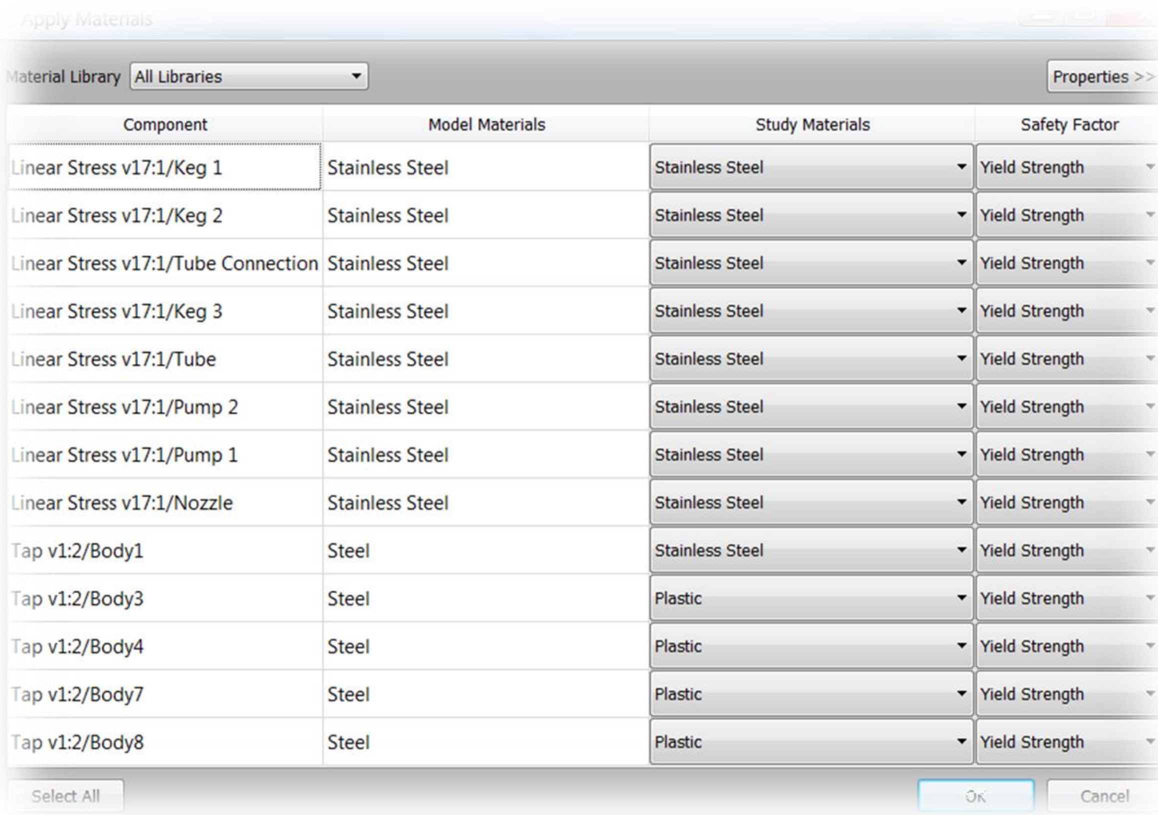
Why: Suppressing these three parts from the analysis will help you to significantly reduce the number of elements created when meshing your model, and therefore speed up your analysis. Since they are not important for the setup, removing these parts will not affect your results.



Define Materials

Step 4: Press **Material** → **Study Material**. Define all parts of the keg to be stainless steel, and the tab handle to be plastic.

Why: Each part in a design needs accurate materials to represent how it will respond under loading conditions. Our area of focus is in the tap handle and connecting points between the handle and keg. Just so long as our parts remain linear, our generic plastic material properties will be acceptable.



Component	Model Materials	Study Materials	Safety Factor
Linear Stress v17:1/Keg 1	Stainless Steel	Stainless Steel	Yield Strength
Linear Stress v17:1/Keg 2	Stainless Steel	Stainless Steel	Yield Strength
Linear Stress v17:1/Tube Connection	Stainless Steel	Stainless Steel	Yield Strength
Linear Stress v17:1/Keg 3	Stainless Steel	Stainless Steel	Yield Strength
Linear Stress v17:1/Tube	Stainless Steel	Stainless Steel	Yield Strength
Linear Stress v17:1/Pump 2	Stainless Steel	Stainless Steel	Yield Strength
Linear Stress v17:1/Pump 1	Stainless Steel	Stainless Steel	Yield Strength
Linear Stress v17:1/Nozzle	Stainless Steel	Stainless Steel	Yield Strength
Tap v1:2/Body1	Steel	Stainless Steel	Yield Strength
Tap v1:2/Body3	Steel	Plastic	Yield Strength
Tap v1:2/Body4	Steel	Plastic	Yield Strength
Tap v1:2/Body7	Steel	Plastic	Yield Strength
Tap v1:2/Body8	Steel	Plastic	Yield Strength

Apply Constraints

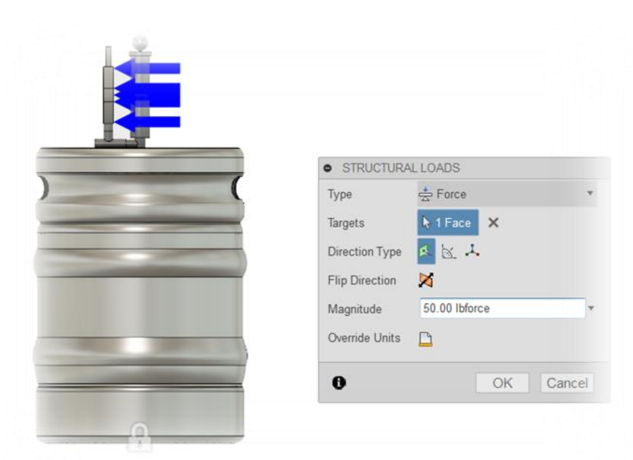
Step 5: Press **Constraint**. Select the surfaces shown in the image below in blue.

Why: We will assume that these surfaces are completely fixed as they are surrounded by ice and the plastic keg tub. Though this is not completely accurate, to simplify a linear stress analysis for worst case conditions, this will be acceptable.

Apply Loads

Step 6: Press **Load → Force.** Select the back surface of the tap handle and apply a 50 pound force.

Why: the 50 pound force applied to the surface will represent us pulling the handle VERY hard. Perhaps we had too much to drink and don't realize our strength. Most parts will have an unrealistic load applied to them and let you know what conditions they have been tested under.

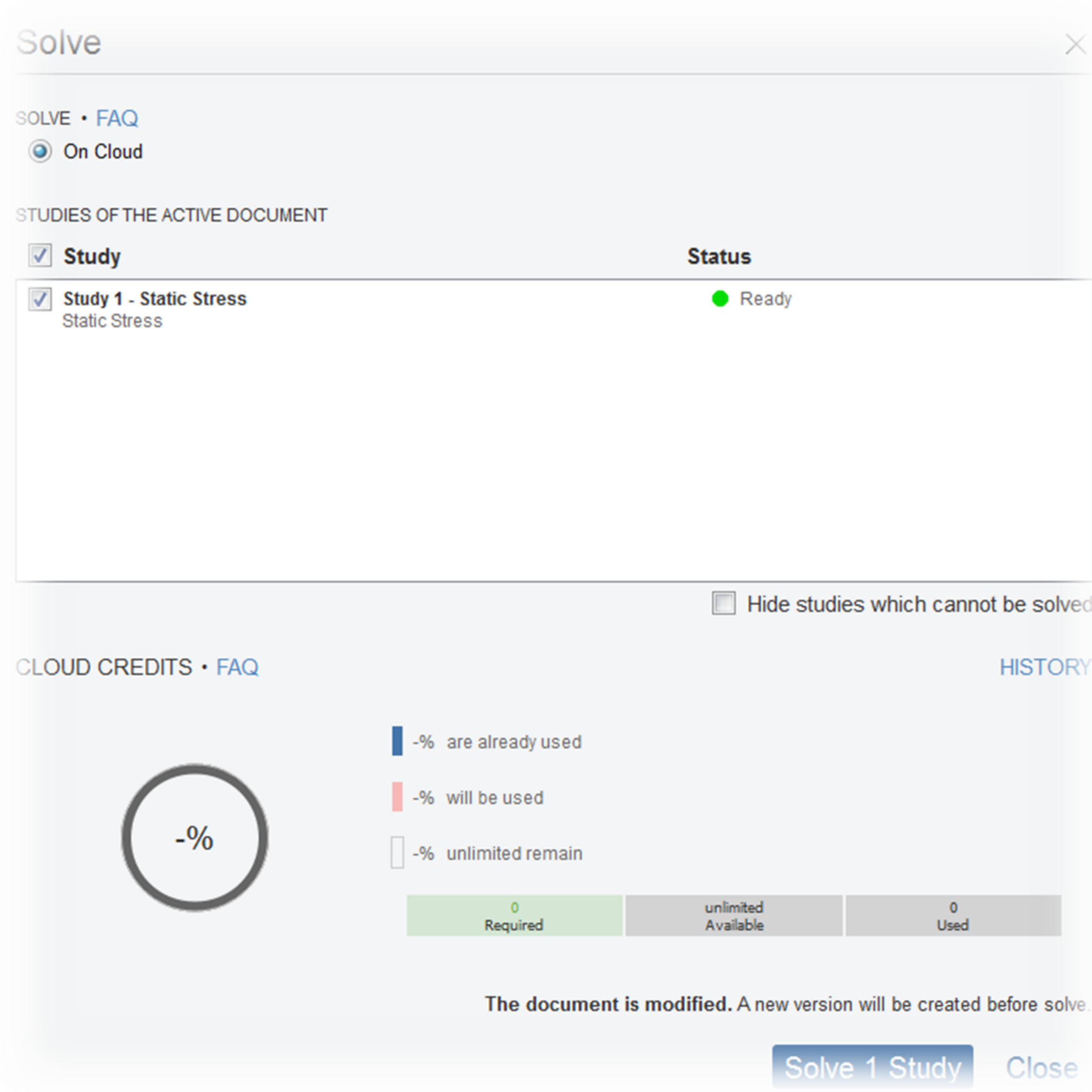


Apply Contact and Solve

Step 7: Press **Contacts → Automatic Contacts → Generate**.

Step 8: Press **Solve → Solve 1 Study**.

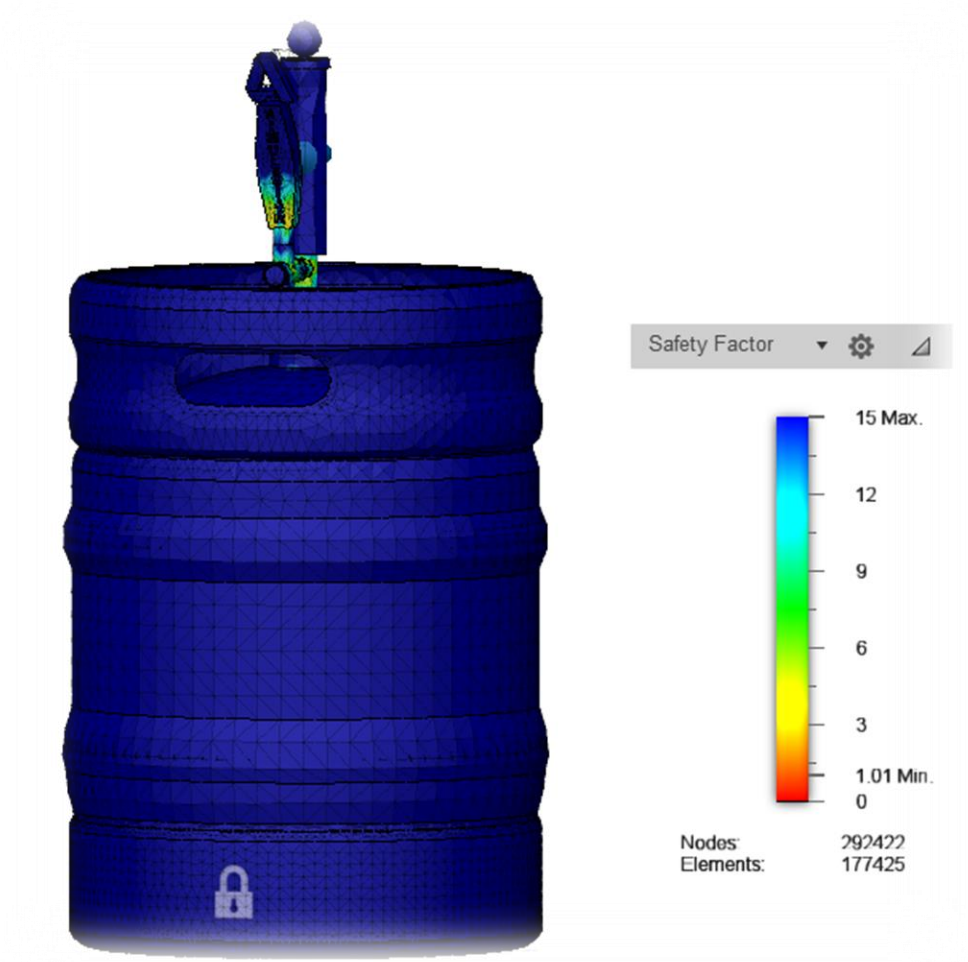
Why: Generating contact will automatically define bonded contact between each part, meaning they are completely tied together. You are able to change this contact definition to surface to surface contact to more realistically represent a designs setup, though if parts will separate this is best done in an event simulation analysis.



Review Results

Step 9: Review your parts safety factor.

Why: Safety factor will tell you if your part will fail (below 1) or pass. As you can see here, the value of 1.01 is very low and would be considered unsafe for use. Reviewing the area of highest stress reveals that the coarse mesh used likely is causing an unrealistic stress value to be calculated in this area.

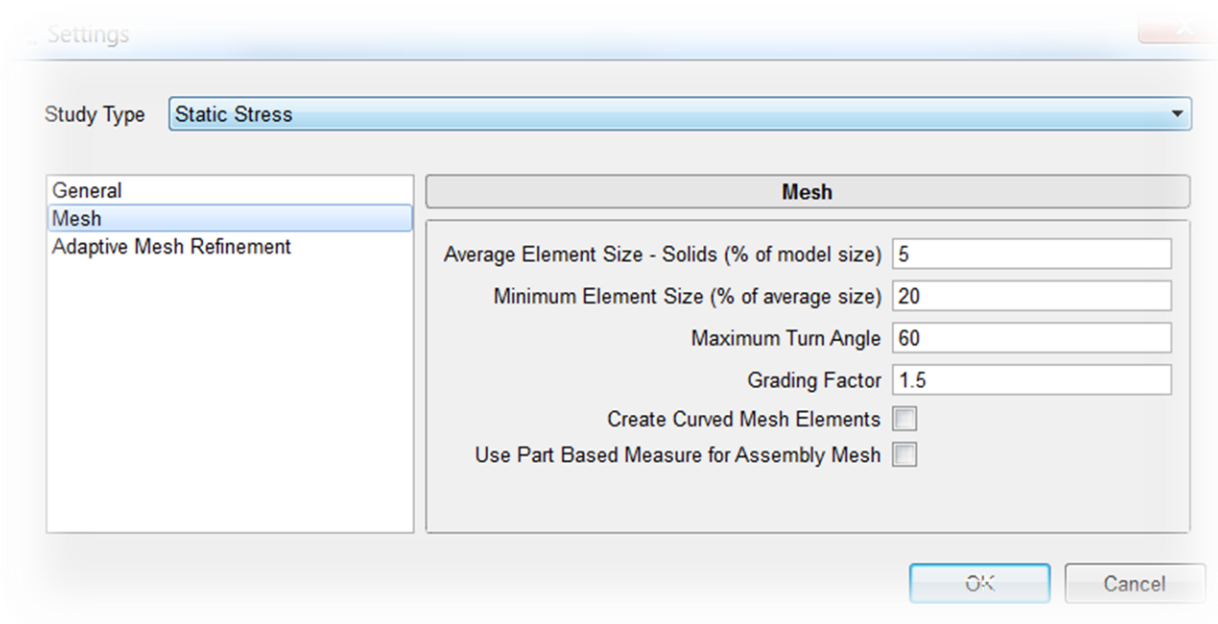


Refine Mesh and Reanalyze

Step 10: Press **Manage → Settings → Mesh.** Change the average element size to 5%.

Step 11: Press **Solve → Solve 1 Study.**

Why: Reducing the mesh size of your assembly will provide smaller elements for the calculations. Although this will lead to longer run times, the results will be more accurate. As you can see, the safety factor rises to 1.47 with the finer mesh, and the large elements in the area of highest stress no longer exist. This provides a more realistic result.



Additional Autodesk Simulation Solutions to this Problem

- Autodesk Inventor Professional
- Autodesk Simulation Mechanical
- Autodesk Nastran In-CAD

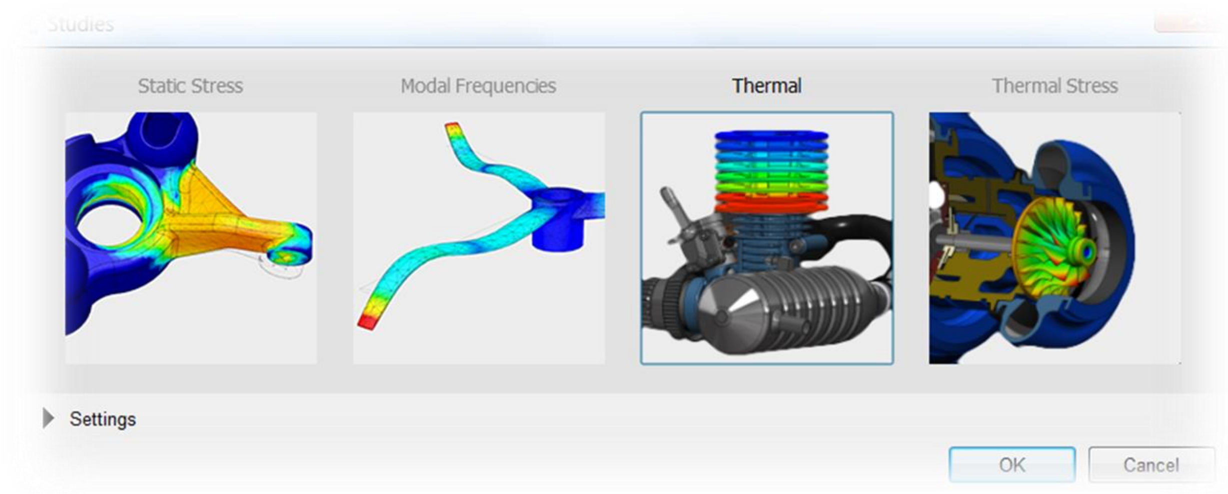
Heat Transfer Analysis

A thermal heat transfer analysis allows you to apply temperature loads such as heat generation, convection and applied temperatures to your model. When analyzed, the program will calculate how these different loads affect the overall assembly temperature in a static condition. It is important to note that this is not a time dependent analysis we are performing, so the results shown are at steady state. Currently Fusion 360 does not allow for a transient (time dependent) thermal analysis, but this can be run in other Autodesk Simulation products. It is also important to note that you must run a thermal stress analysis in order to see how the change in temperature affects stress in your design.

Change the Analysis Type

Step 1: Press **Study** → **Thermal**

Why: Choosing the thermal analysis will allow you to understand how your parts will heat up under various thermal loading conditions. To see how these changes in temperature affect stress results, a Thermal Stress analysis must be performed.



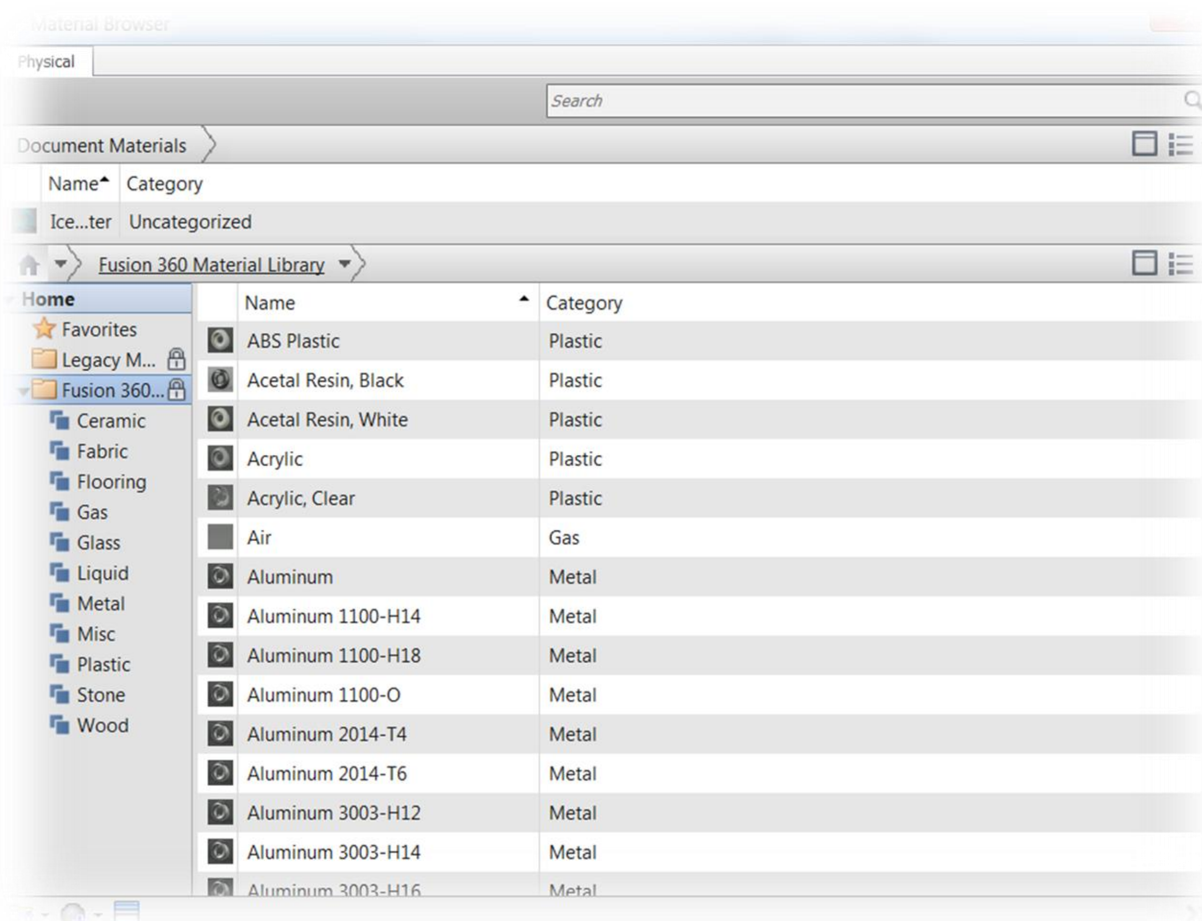
Create a Custom Material

Step 2: Press **Material → Manage Physical Properties**

Step 3: Press **Create → New Library**

Step 4: Press **Create → New Material**

Why: There are times where the material that you need is not available in the default material library (in this case, ice water). Creating a custom material will define your parts more accurately, giving you more accurate results.



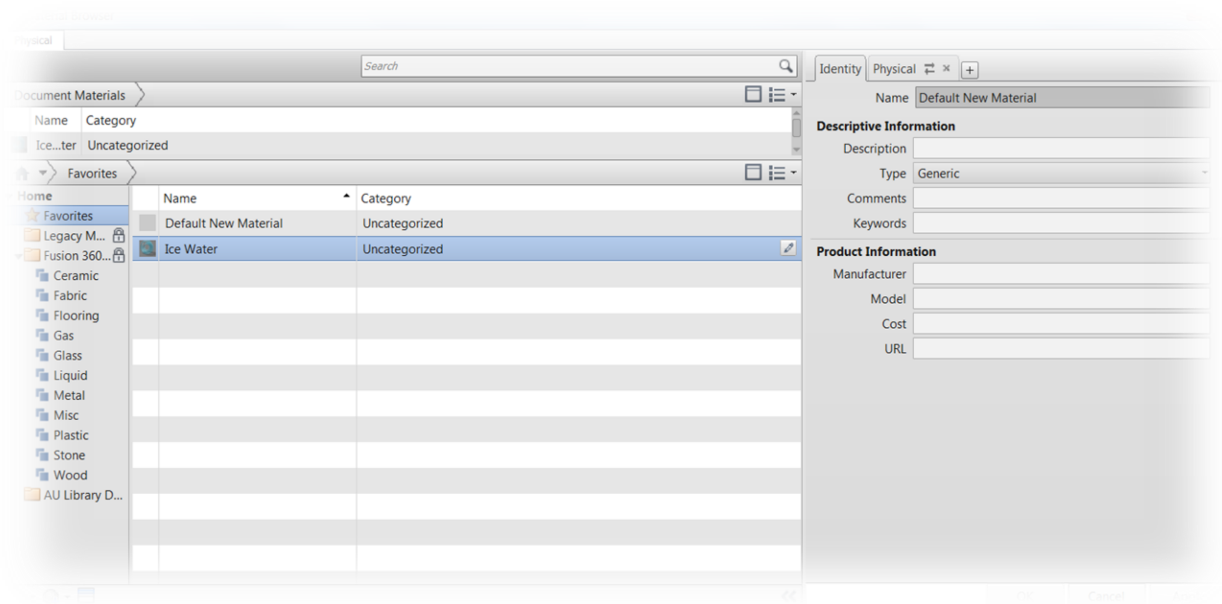
Create a Custom Material (cont.)

Step 5: Provide a material name and description.

Step 6: Press **+ Physical Properties**

Step 7: Define Thermal Conductivity and Specific Heat. Press Apply.

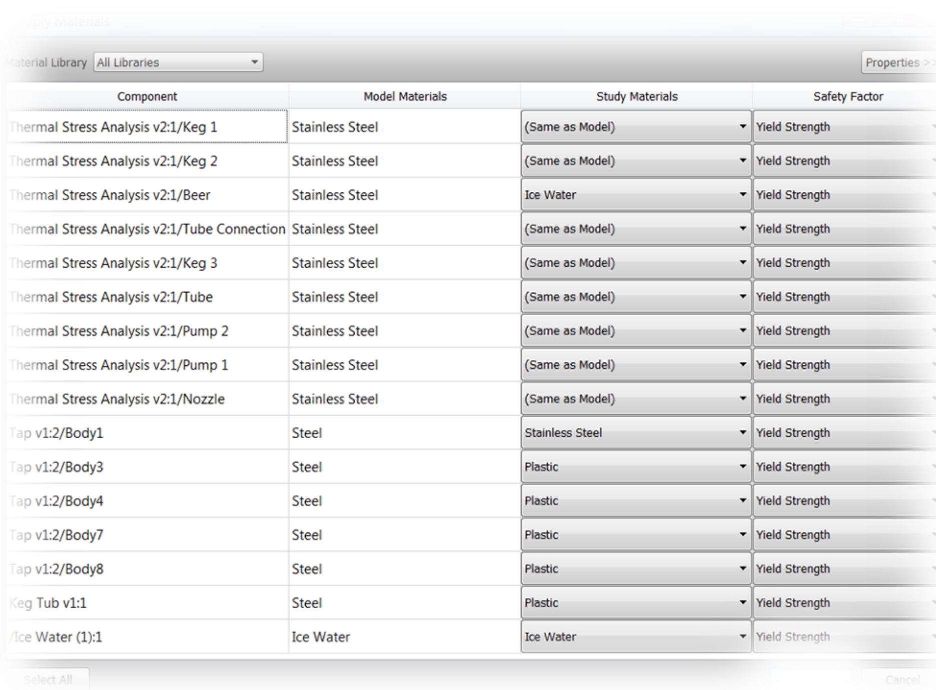
Why: Since we are running a thermal analysis, thermal conductivity and specific heat are the material properties of need. This same process can be performed to add new physical properties for stress analyses as well.



Define Materials

Step 8: Press Material → Study Materials. Define the materials as shown in the image below.

Why: Steel, plastic and water all have significantly different material properties. In order to have accurate heat transfer between these parts, defining each with the proper material is necessary. Although ice water is a liquid/solid mix, we are assuming it to be a solid part with consistent material properties as part of the analysis simplification.

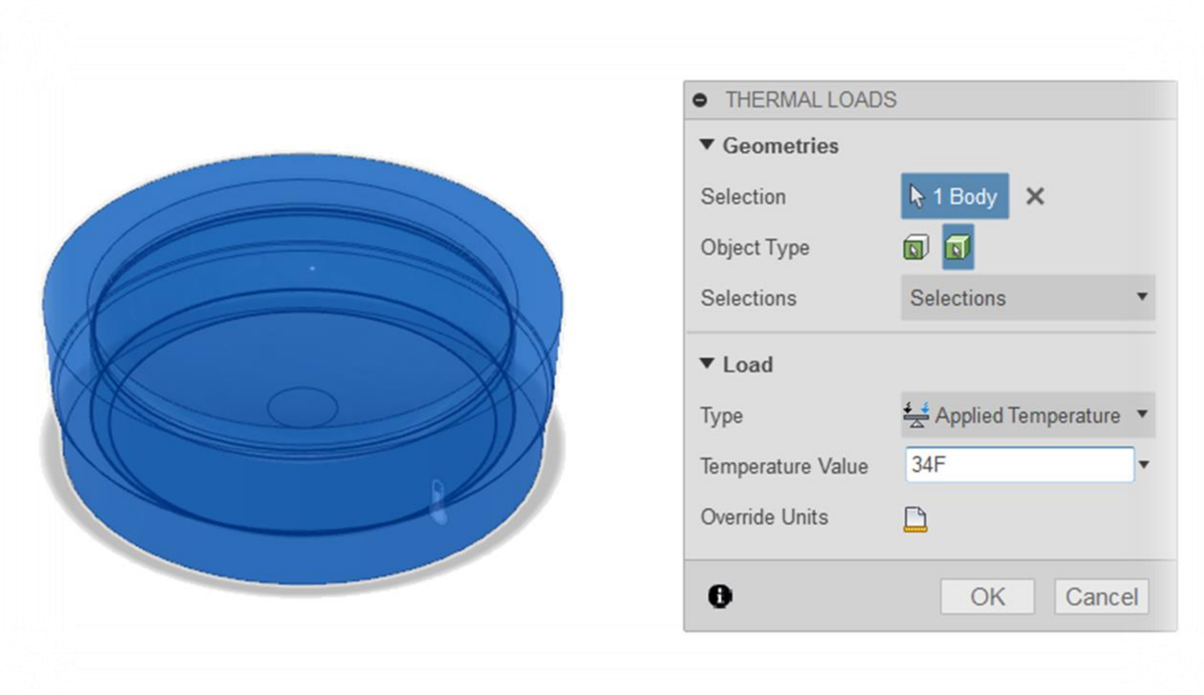

Material Library All Libraries Properties >>

Component	Model Materials	Study Materials	Safety Factor
Thermal Stress Analysis v2:1/Keg 1	Stainless Steel	(Same as Model)	Yield Strength
Thermal Stress Analysis v2:1/Keg 2	Stainless Steel	(Same as Model)	Yield Strength
Thermal Stress Analysis v2:1/Beer	Stainless Steel	Ice Water	Yield Strength
Thermal Stress Analysis v2:1/Tube Connection	Stainless Steel	(Same as Model)	Yield Strength
Thermal Stress Analysis v2:1/Keg 3	Stainless Steel	(Same as Model)	Yield Strength
Thermal Stress Analysis v2:1/Tube	Stainless Steel	(Same as Model)	Yield Strength
Thermal Stress Analysis v2:1/Pump 2	Stainless Steel	(Same as Model)	Yield Strength
Thermal Stress Analysis v2:1/Pump 1	Stainless Steel	(Same as Model)	Yield Strength
Thermal Stress Analysis v2:1/Nozzle	Stainless Steel	(Same as Model)	Yield Strength
Tap v1:2/Body1	Steel	Stainless Steel	Yield Strength
Tap v1:2/Body3	Steel	Plastic	Yield Strength
Tap v1:2/Body4	Steel	Plastic	Yield Strength
Tap v1:2/Body7	Steel	Plastic	Yield Strength
Tap v1:2/Body8	Steel	Plastic	Yield Strength
Keg Tub v1:1	Steel	Plastic	Yield Strength
Ice Water (1):1	Ice Water	Ice Water	Yield Strength

Define Load – Applied Temperature

Step 9: Press Load → Applied Temperature → 34F. Select all surfaces of the ice water and click OK.

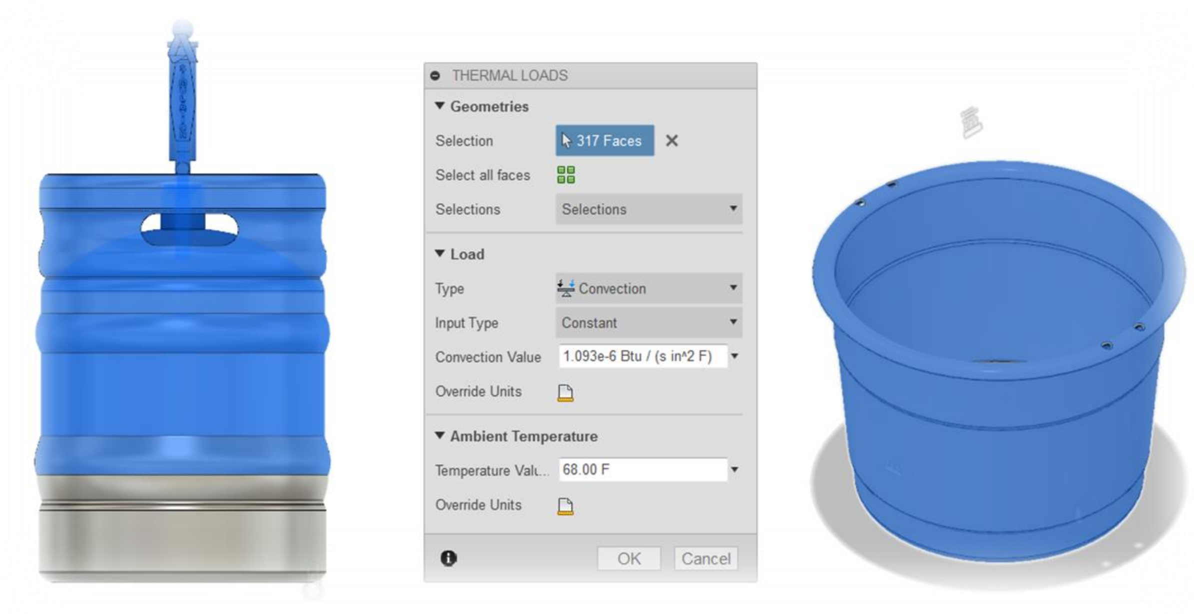
Why: Ice water is slightly above freezing, so an estimated value of 34F is used. For a thermal analysis to run successfully, you need at least 2 loading conditions. This value will act as our “temperature in” for the model, with the next convection load as the “temperature out.” You are also able to apply this load to the body of the ice water instead of selecting surfaces.



Define Load – Convection

Step 10: Press Load → Convection → 1.093e-6 Btu/(s in^2 F). Select the surfaces shown in blue in the image below. After defining the convection coefficient, it is important to define the ambient temperature to be at room temperature, or 68F.

Why: The convection coefficient defines how heat will transfer from the ice to our keg, as well as state the outside temperature surrounding the parts to be 68F. This value can be calculated using scary college thermodynamics.

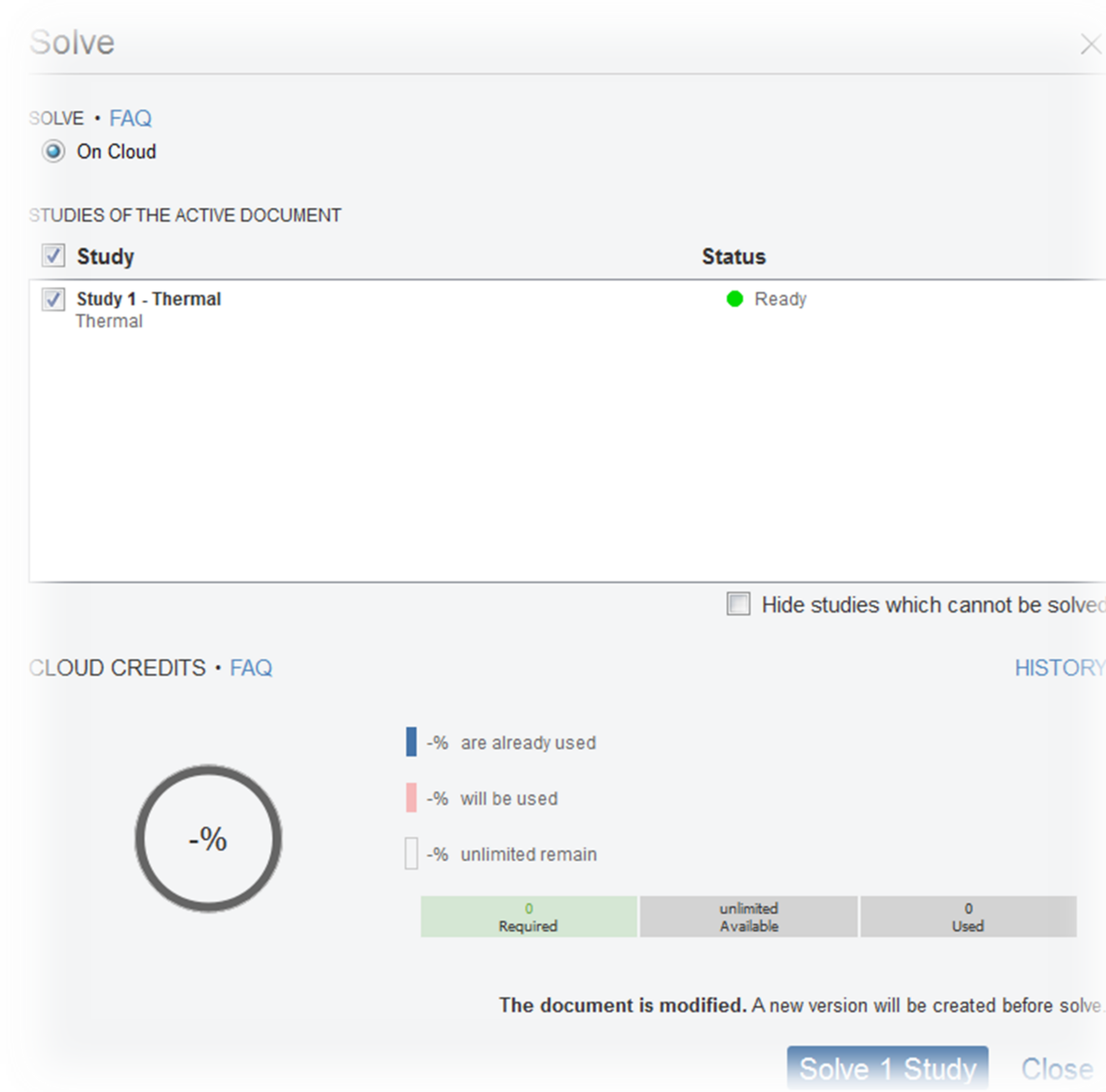


Define Contact and Solve

Step 11: Press Contact → Automatic Contact → Generate

Step 12: Press Solve → Solve 1 Study

Why: Defining contact between parts in a thermal analysis determines if parts are conducting on insulating. As was the case with linear stress, the default contact setting is bonded. When entering the contact definition screen you are able to define a resistance value between surfaces. This is useful if there is an epoxy connecting parts as one example.

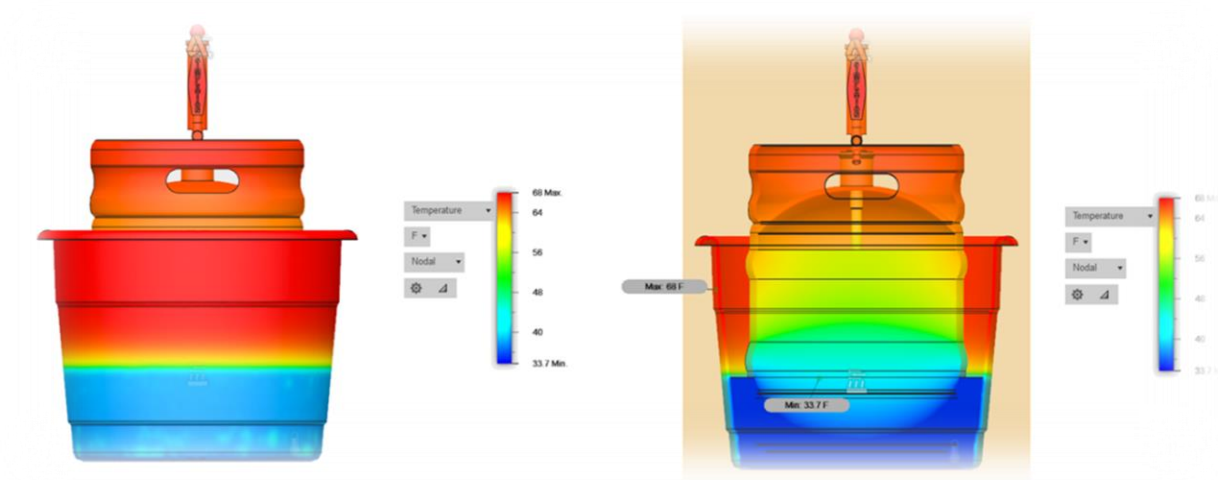


Define Contact and Solve

Step 13: Review the temperature results.

Step 14: Press Inspect → Create Slice Plane. Choose a flat surface to apply the slice plane to and press OK.

Why: Applying a slice plane to your results will help you to understand the temperature distribution to interior parts, in this case the beer. This will tell us if the beer is cold enough or if more ice needs to be added.



Additional Autodesk Simulation Solutions to this Problem

- Autodesk Simulation Mechanical
- Autodesk Nastran In-CAD
- Autodesk CFD

Event Simulation

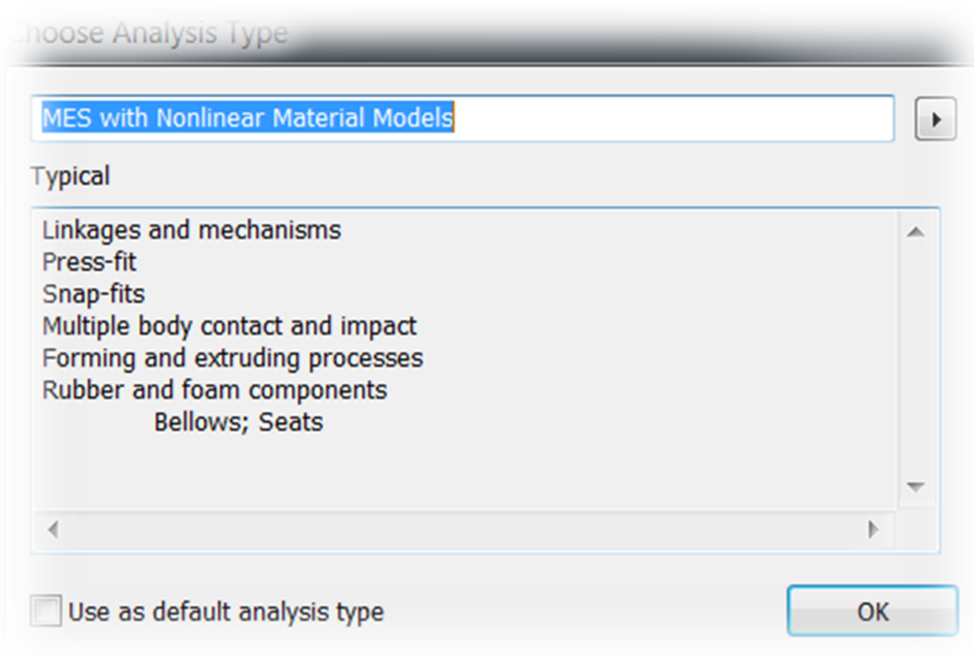
The term event simulation is rather general, describing basically any analysis where there is motion involved. One of the most common cases for using event simulation is for a drop test analysis. Although this can be a very long analysis, and computationally significant, the cost savings in the long run from reducing prototypes and test equipment can be huge! Event simulation analyses can be some of the most real world applications for simulation, allowing parts to come into contact with each other at any time during the analysis. The addition of movement and time based results can allow you to see how parts will react when acted upon by another force or part. In this case, we will look at how our keg bounces after dropping off of the ground.

Autodesk Simulation Mechanical Workflow

Choose the Analysis Type

Step 1: Choose Mechanical Event Simulation

Why: Autodesk Simulation Mechanical allows for larger models to be analyzed in drop test scenarios. This means you can see more of the reaction your design has after impact, as well as stress results from the contact.

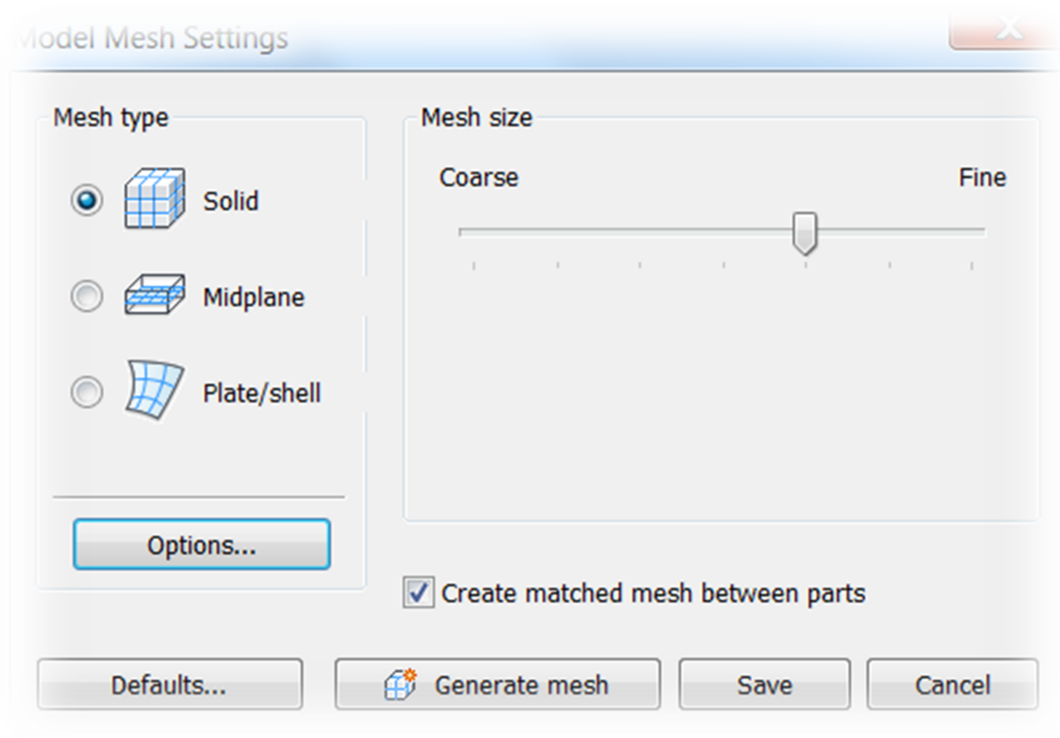


Mesh the Assembly

Step 2: Choose Mesh → 3D Mesh Settings → Options → Model and choose to uncheck "Use automatic geometry-based mesh size function."

Step 3: Set Mesh Size to 70% → Mesh

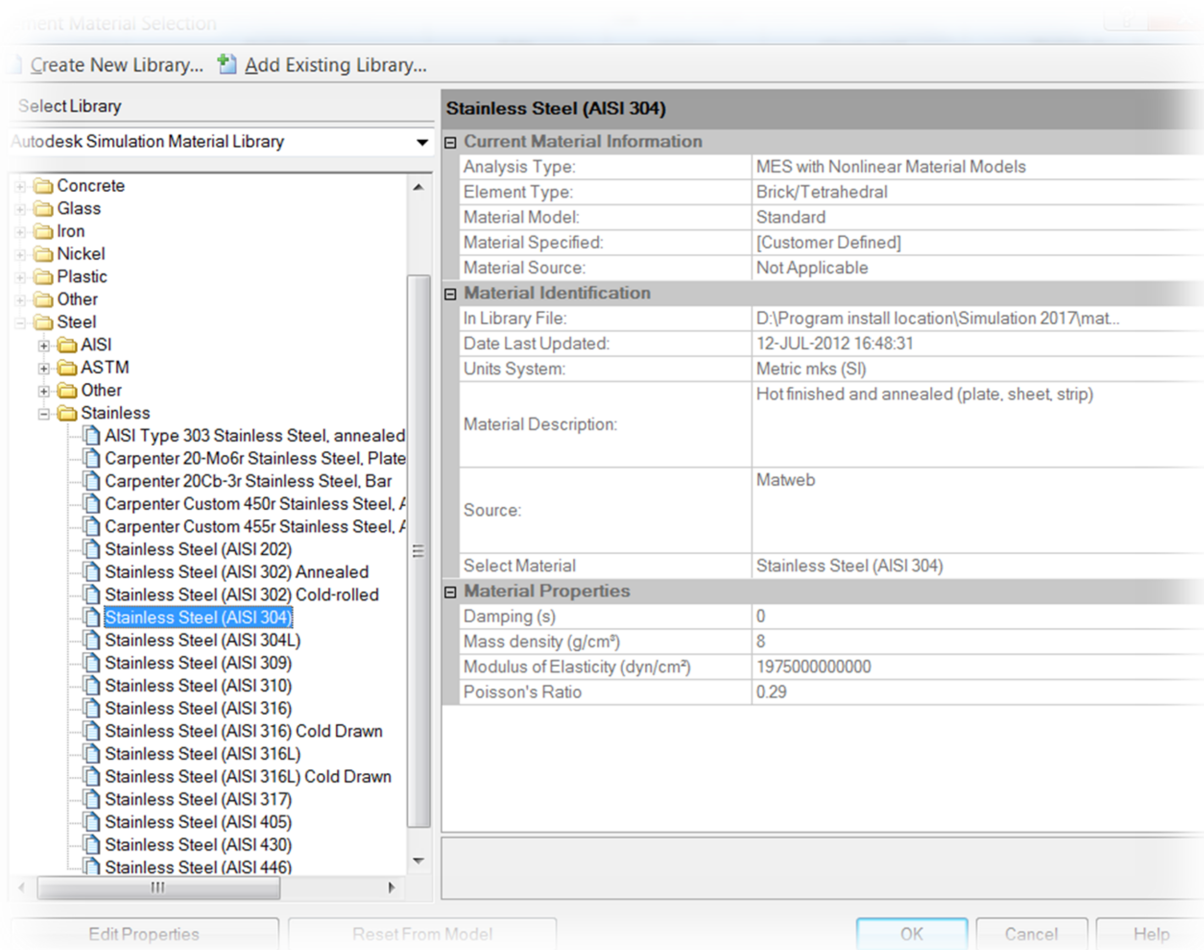
Why: Deactivating geometry-based mesh sizing allows for larger elements to be used when meshing. This means that the analysis will run more quickly. If you find your part needs a finer mesh, using local mesh refinement is a better option to keep element count low.



Define Materials

Step 4: Select Material for each part in the model tree, right click and choose edit material. Select 304 stainless steel and press OK.

Why: Before importing the model it is important to simplify it by suppressing parts that are not critical for the analysis. In this case, we only have 4 parts remaining for the drop test analysis, all of which are created out of stainless steel.



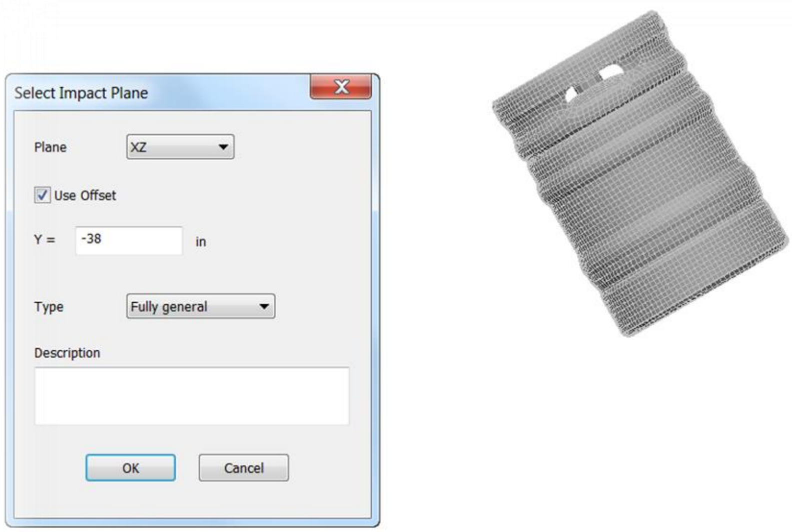
Create an Impact Plane

Step 5: Press Selection → Point → Vertices. Choose the bottom point on the keg and inquire.

Step 6: Press Setup → New Impact Plane. Define the plane to be in the XZ plane, use offset of -38.6 inches in the Y direction. Change the Type to fully general.

Why: Using an impact plane instead of modeling another “floor” part in the model saves on elements and helps to run more quickly. To make sure your part is falling the right distance, inquiring on the bottom most point will help you determine where to define your impact plane. Using the fully general option will allow you to see your design bounce after impact, a more intensive analysis but most realistic.

C



Define Analysis Parameters

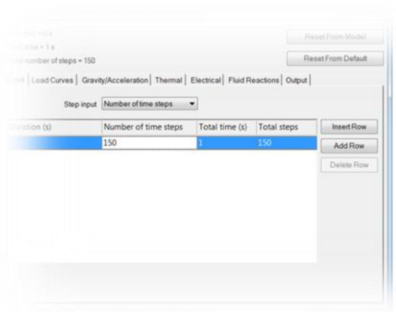
Step 7: Press Setup → Model Setup → Parameters.

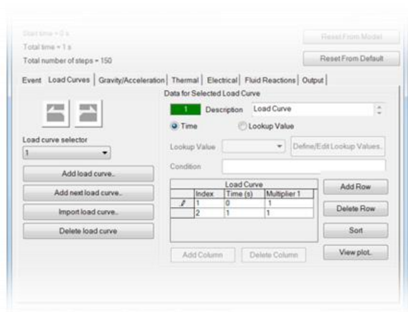
Step 8: Press Event Tab → Number of time steps = 150.

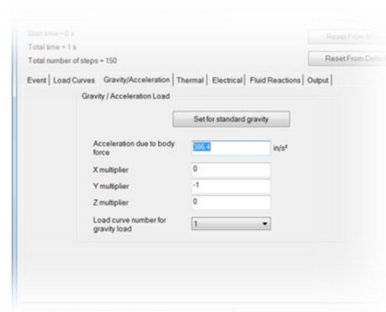
Step 9: Press Load Curves Tab, define the multipliers table as shown below.

Step 10: Press Gravity Tab → Set for standard gravity. Define gravity in the –Y direction as shown below and press OK.

Why: In the analysis parameters is where you define how your loads will be applied over time, as well as how many time steps your analysis is broken into. Since we are running a drop test, gravity needs to be active at all times which is why our multiplier is set to 1 for the entire analysis. We use 150 time steps to help break the calculations into smaller parts, helping to ensure contact between the impact plane and keg is found as well as helping to analyze more quickly.







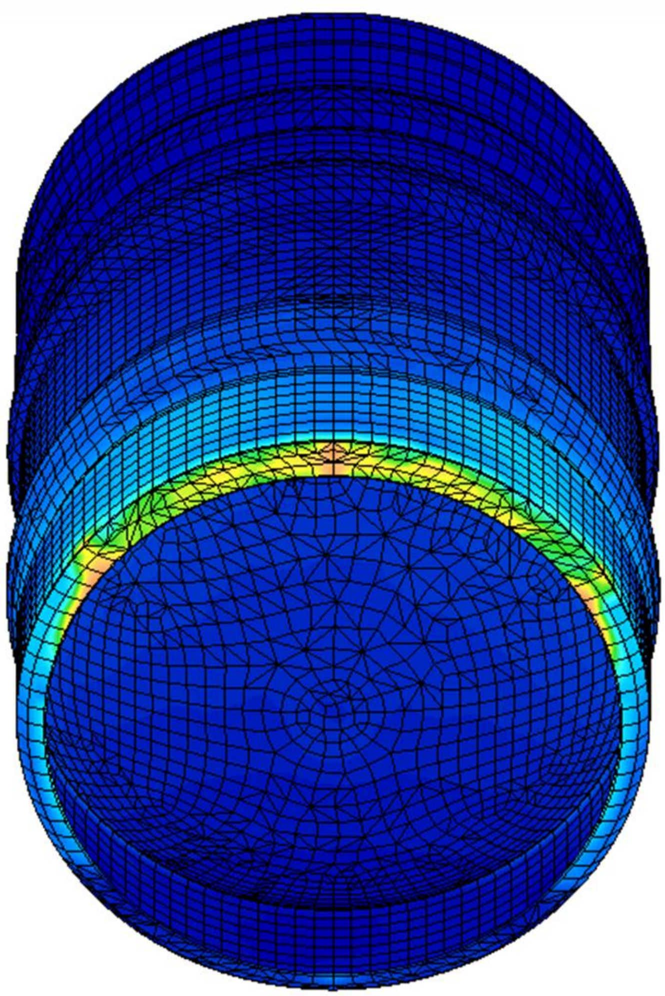
Solve and Review Results

Step 11: Press Analysis → Run Simulation.

Step 12: Press Load Case Options → Load Case → Set = 115.

Step 13: Press Results Contours → Stress → von Mises Stress.

Why: To be able to see stress results at the worst time of the analysis you have to locate when your design makes contact with the impact plane. Changing to von Mises stress will let you know if your part will fail based on the material properties. In this case, we're ok!

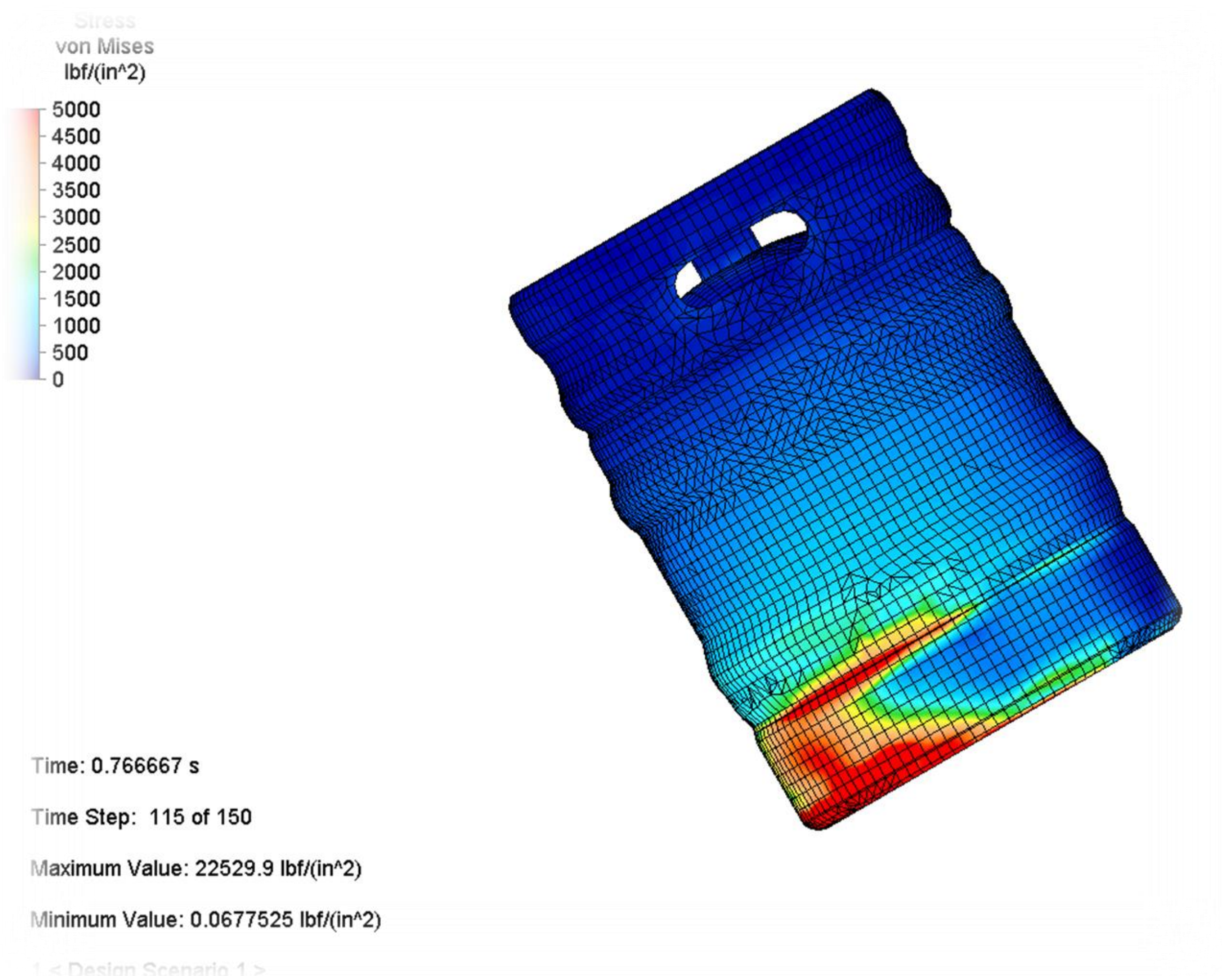


Modify Legend Properties

Step 14: Press Results Contours → Legend Properties → Range and uncheck "automatically calculate value range".

Step 15: Define Low = 0 and High = 5,000. Press Apply.

Why: With a lower value for the high stress on our legend it is easier to see areas of high stress. In this case the stress values are localized, but in many analyses you will have high stress in multiple areas and this can help you pin point problems in your design.



Additional Autodesk Simulation Solutions to this Problem

- Autodesk Nastran In-CAD
- Autodesk Fusion 360

Injection Location and Visualization Analysis

A design is able to help cut costs by recognizing problems early on in the design cycle. When working with plastic parts, having the ability to optimize injection site locations and visualize defects on the fly can help a designer to understand if changes need made before moving on to further analysis or manufacturing. In addition, gauges for manufacturability and eco-friendly controls guide designers in material choice and part shape to improve process flow in the manufacturing process.

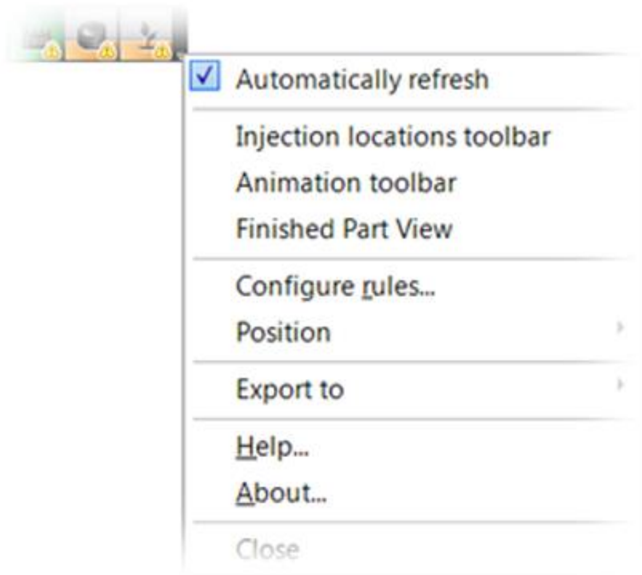
Autodesk Moldflow Design Workflow

Add Injection Site Location

Step 1: Click dropdown → Injection locations toolbar.

Step 2: Click Add injection location. Choose two injection locations on the inside of the keg tub near the handles as shown below.

Why: By default there will be 1 injection location applied to the part. Depending on the part, there are times where multiple injection locations would be better for the molding process.



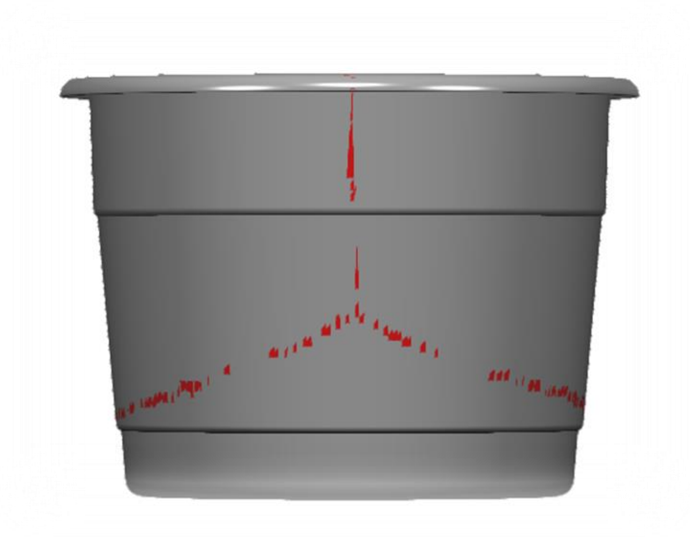
Visualize Defects

Step 3: Click dropdown → Animation toolbar. Press the play button to watch the fill process.

Step 4: Click dropdown → Finished part view.

Step 5: Click View → Highlight defects.

Why: After visualizing defects in the part you can determine where sink marks and weld lines will occur. Not only will this help to design a stronger part, as weld lines will weaken the design, but it will also help to create a more aesthetically pleasing design. Often times smaller sink marks are able to be hidden by material color.



Optimize Injection Location

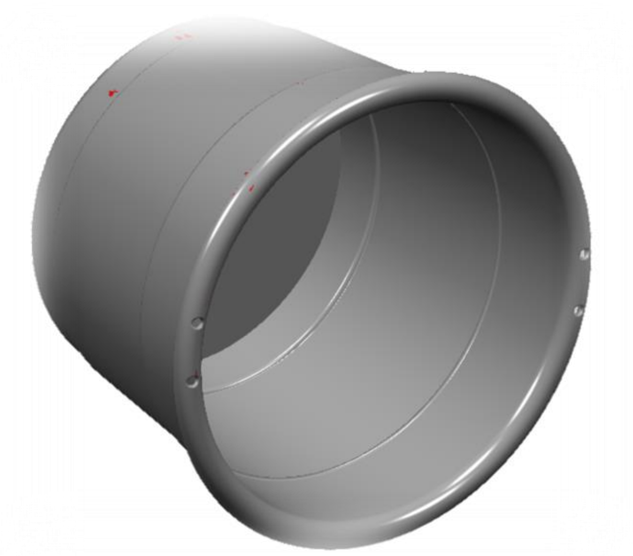
Step 6: Click Delete injection location. Remove the 2 previously added injection site locations.

Step 7: Press the animation button to watch the new fill process.

Step 8: Click dropdown → Finished part view.

Step 9: Click View → Highlight defects.

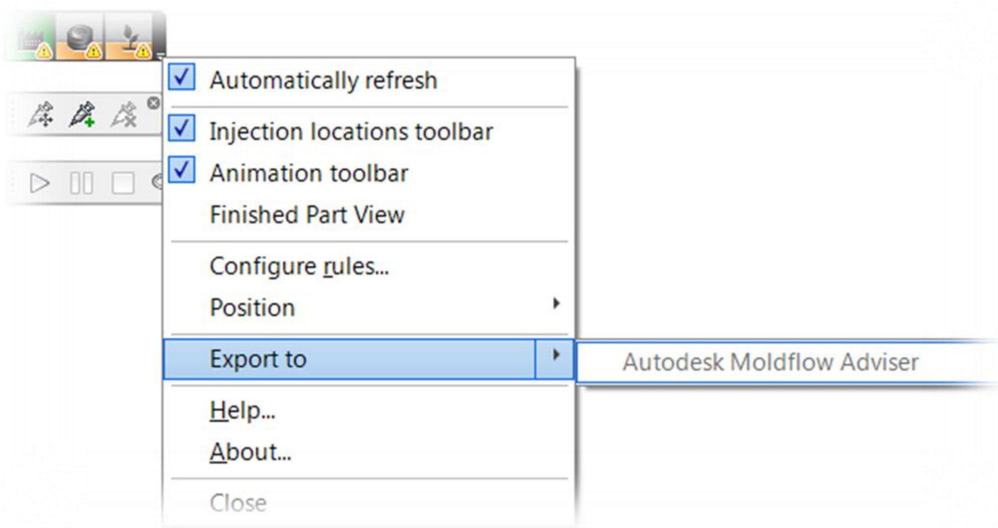
Why: With only 1 injection location the weld lines that were being created no longer exist. Now there are just a few sink marks that could be hidden with material color.



Export to Moldflow

Step 10: Click the dropdown → Export to → Autodesk Moldflow.

Why: By exporting to Moldlfow your selected injection locations and material choices will be saved and you will be able to run more complex analyses on the design.



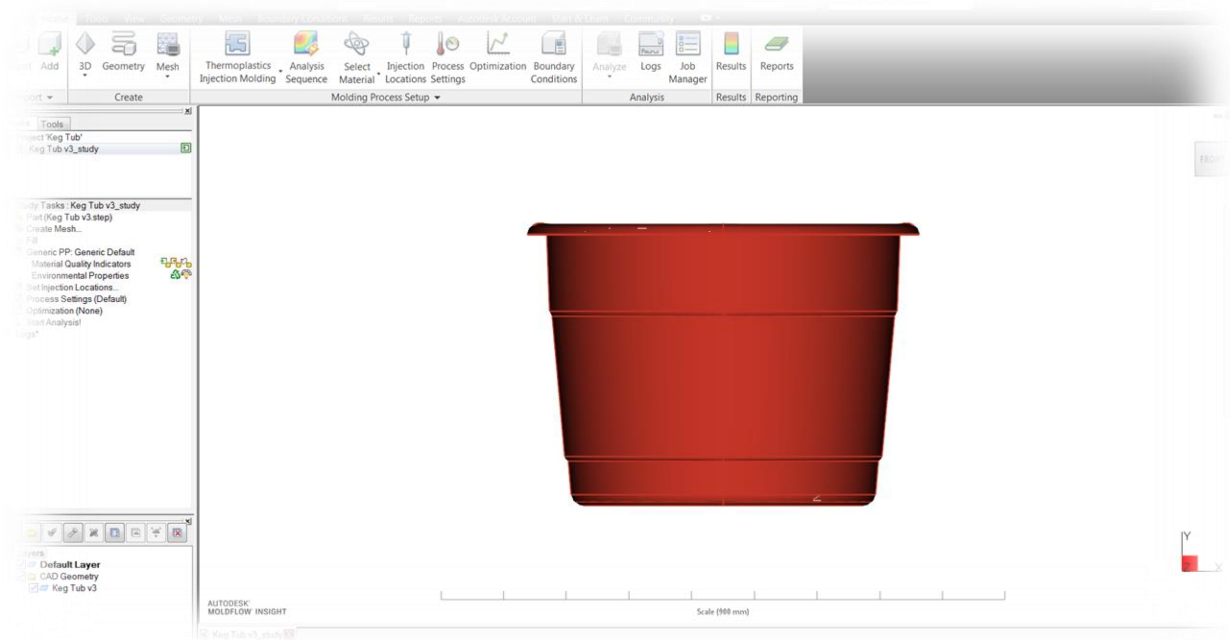
Additional Autodesk Simulation Solutions to this Problem

- Autodesk Moldflow Adviser
- Autodesk Modlfow Insight

Fill / Pack / Warp Analysis

A common analysis set run on plastic injection molded parts is the Fill + Pack + Warp analysis process. In addition to understanding if your location site will work for your design, you will be able to determine how long it will take for your part to fill, as well as if any warping of the material will occur during the cooling process. With this you are able to optimize material use, develop stronger parts based on flow lines and make sure that the finished part is sized within the necessary limits for the project.

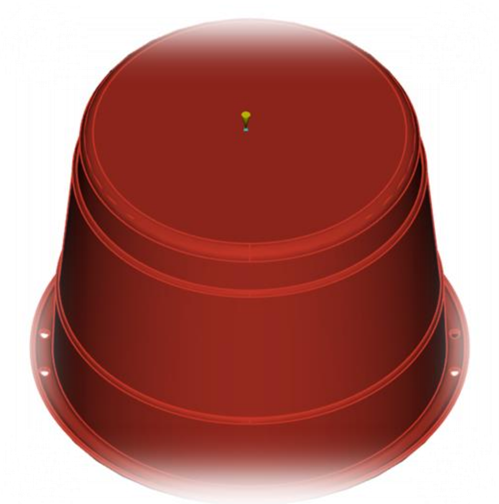
Autodesk Moldflow Workflow



Add Injection Location

Step 1: Click Home → Molding Process Setup → Injection Locations.

Why: If your design was not sent over from Moldflow Design, it is necessary to apply an injection location. We chose the bottom of the tub to help hide the mark left from the gate.

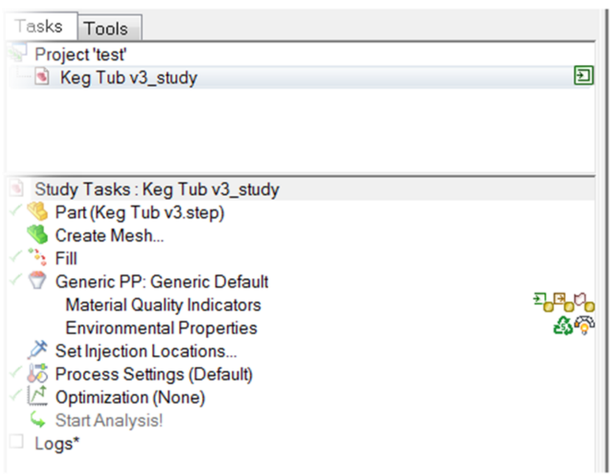


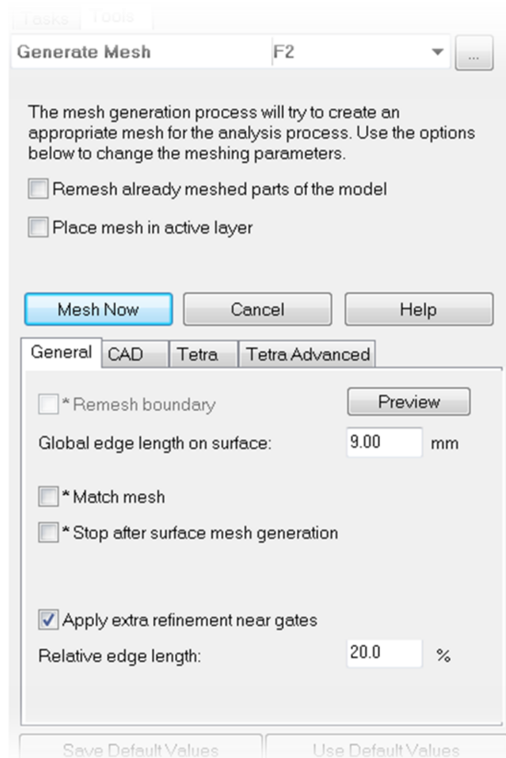
Apply Mesh

Step 2: Double click Create Mesh in Tasks.

Step 3: Global edge length = 9mm, uncheck match mesh, apply extra refinement near gates and click Mesh Now.

Why: It is important to have at least 6 – 8 layers of elements through the thickness of non-fiber filled parts, and mesh refinement is especially important around gate locations to analyze proper flow.

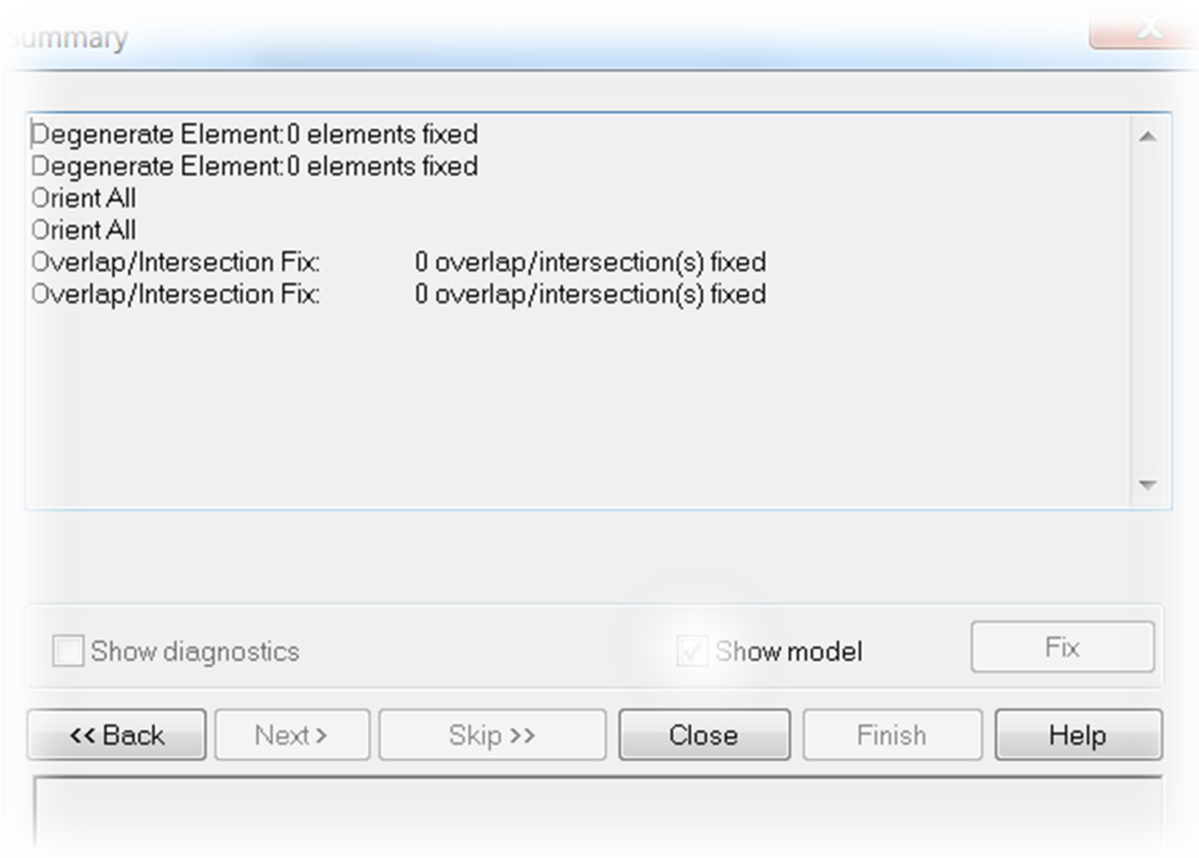




Repair Mesh

Step 4: Choose Mesh → Mesh Repair Wizard. Choose to fix any problems in the mesh.

Why: Since having a good mesh is extremely important, Moldflow provides a built in mesh repair wizard. This should always be used to ensure you have the best mesh possible on your parts, which will in turn provide you with more accurate results.

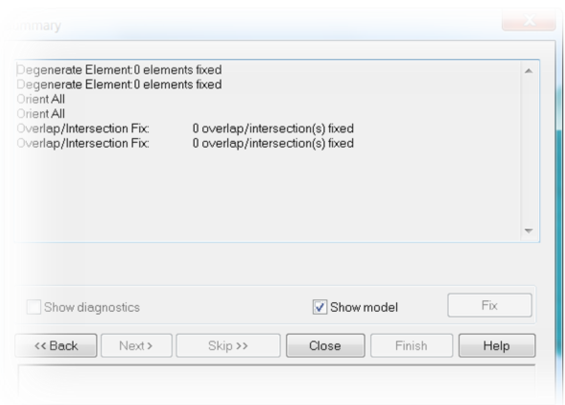


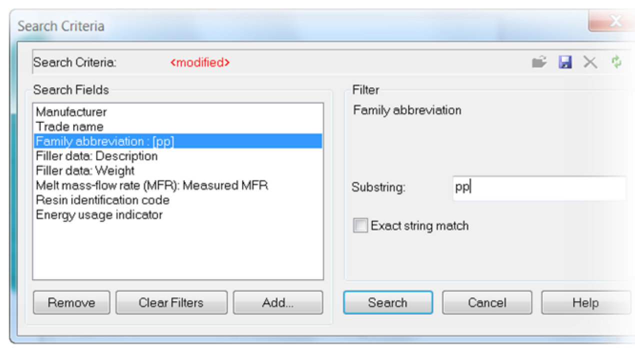
Define Materials

Step 5: In Tasks, double click on Generic PP → Generic Default.

Step 6: Click Search → Family Abbreviation → PP and Manufacturer → Generic. Select Generic Default and press OK.

Why: Moldflow provides nearly 10,000 different materials for you to choose from in the supplied libraries. This often times makes it necessary to use the search functionality to narrow down the materials to what you are interested in.

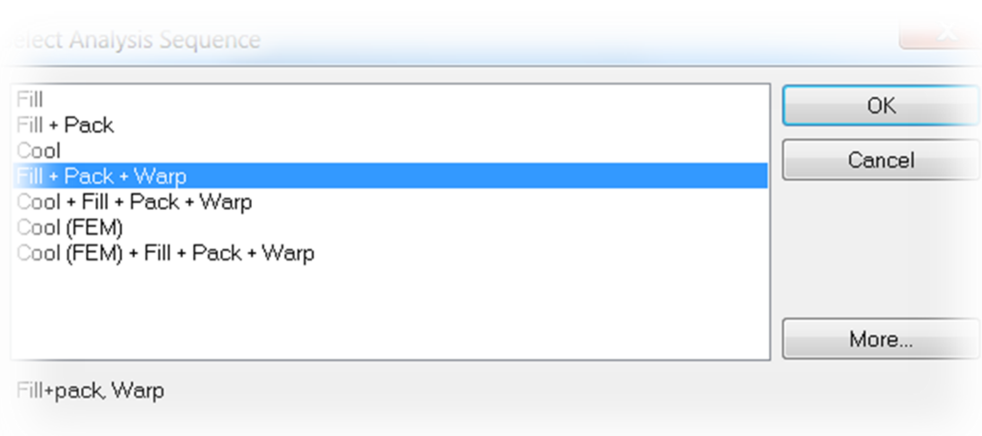




Select Analysis Sequence

Step 7: In Tasks, double click on Fill. Choose Fill + Pack + Warp and press OK.

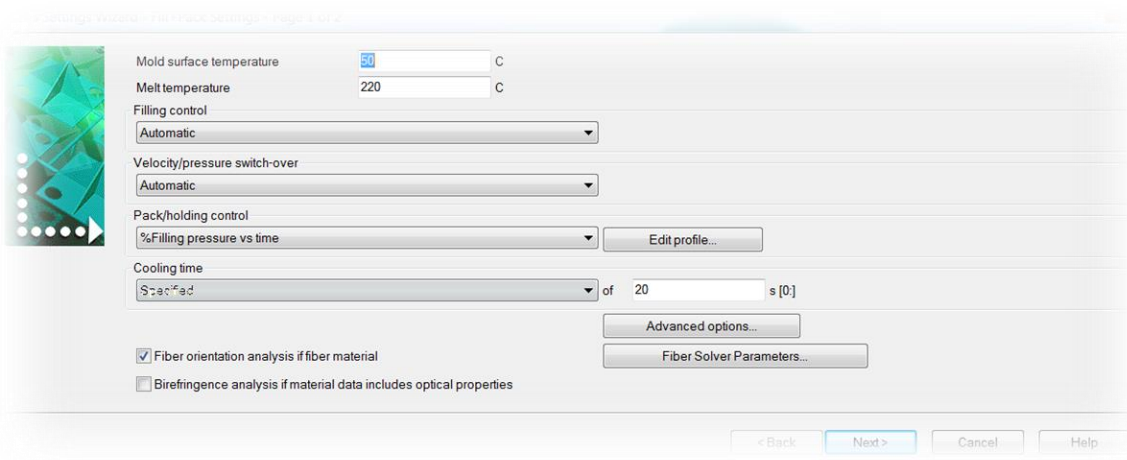
Why: To get more information about the design process, the Fill + Pack + Warp sequence will help you to determine if your part will be created in the best fashion possible. It is important to note that the of the analyses does matter, and various options are available for you to choose from depending on your specific setup.



Define Process Settings

Step 7: In Tasks, double click on Process Settings. Click Next and Finished.

Why: Default process settings are usually good to give you a general understanding of how your part will be filled and how much it will warp. You are able to modify parameters such as mold temperature and melt temperature based on your setup.

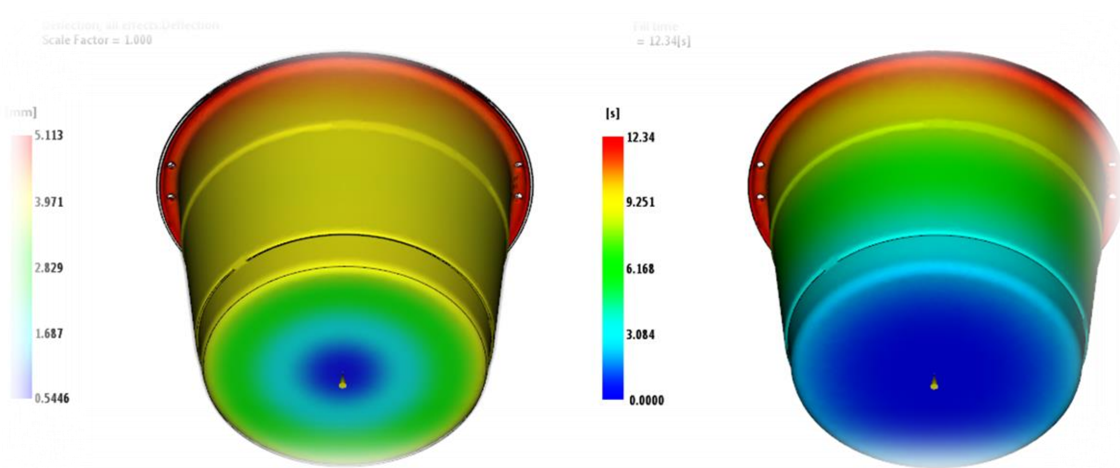


Solve and Review Results

Step 8: In Tasks, double click Start Analysis.

Step 9: Check the box next to Deflections → all effects → Deflections.

Why: As shown below, results such as fill time and deflection can be visualized. It can be unclear as to how much and in which direction your part is warping, but by modifying the scale of deflection in your results options you can more easily visualize the warp results.



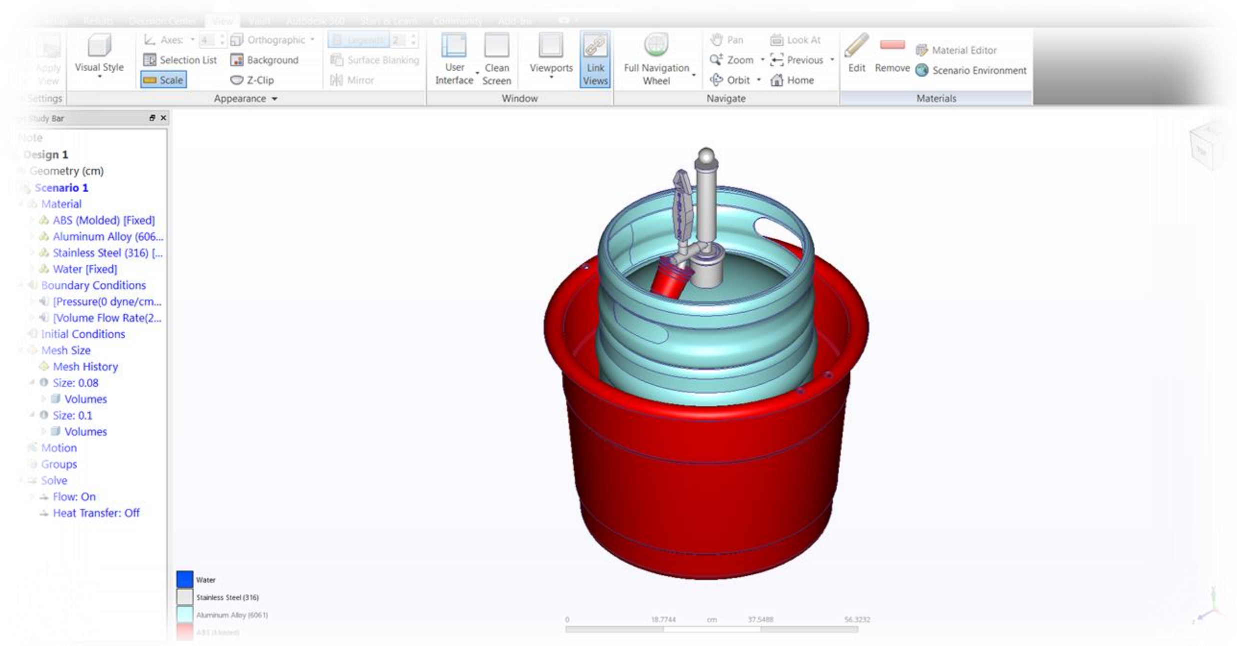
Additional Autodesk Simulation Solutions to this Problem

- Autodesk Moldflow Adviser
- Autodesk Moldflow Insight

Free Surface Flow Analysis

Computational fluid dynamics (CFD) can solve any number of problems ranging from heat transfer to fluid flow, as well as coupling the two analyses to understand how changes in temperature can affect flow. In this analysis we will focus on one of the more intense analyses available, free surface flow. The results of this analysis will helps one to understand how fluid in a contained system reacts after it leaves that system and is transferred into another container, allowing for optimization in pressure to provide the best flow rate possible. This is a very heavy analysis and can take a significant amount of time to solve.

Autodesk CFD Workflow



Define Materials

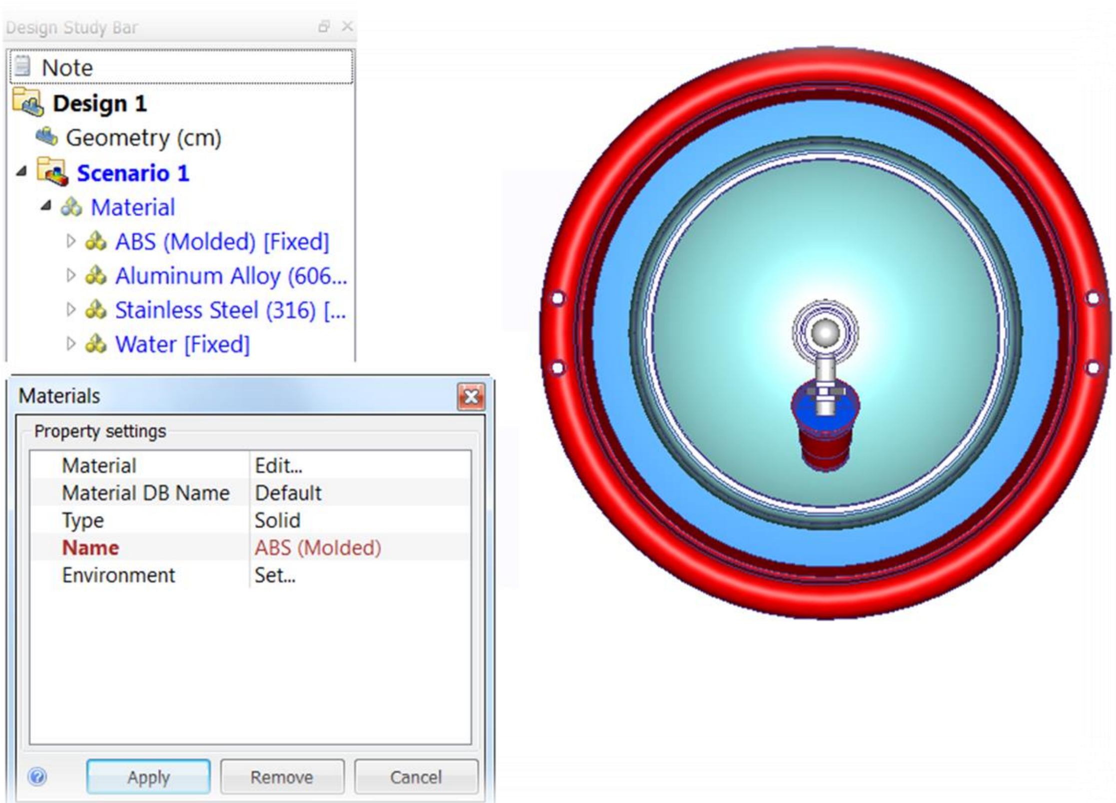
Step 1: Choose the 3 main parts of the keg body and change Type → Solid → Aluminum Alloy (6061).

Step 2: Choose the ice and beer and define Type → Fluid → Water.

Step 3: Choose the cup and keg tub and define Type → Solid → ABS (Molded).

Step 4: Choose the remaining parts and define Type → Solid → Stainless Steel (316).

Why: It is extremely important to properly define materials, especially the "Type." If the beer is not set to fluid, we will not be able to see if flow out of the keg. Since temperature isn't being taken into account in this analysis, the other material properties are not as important.

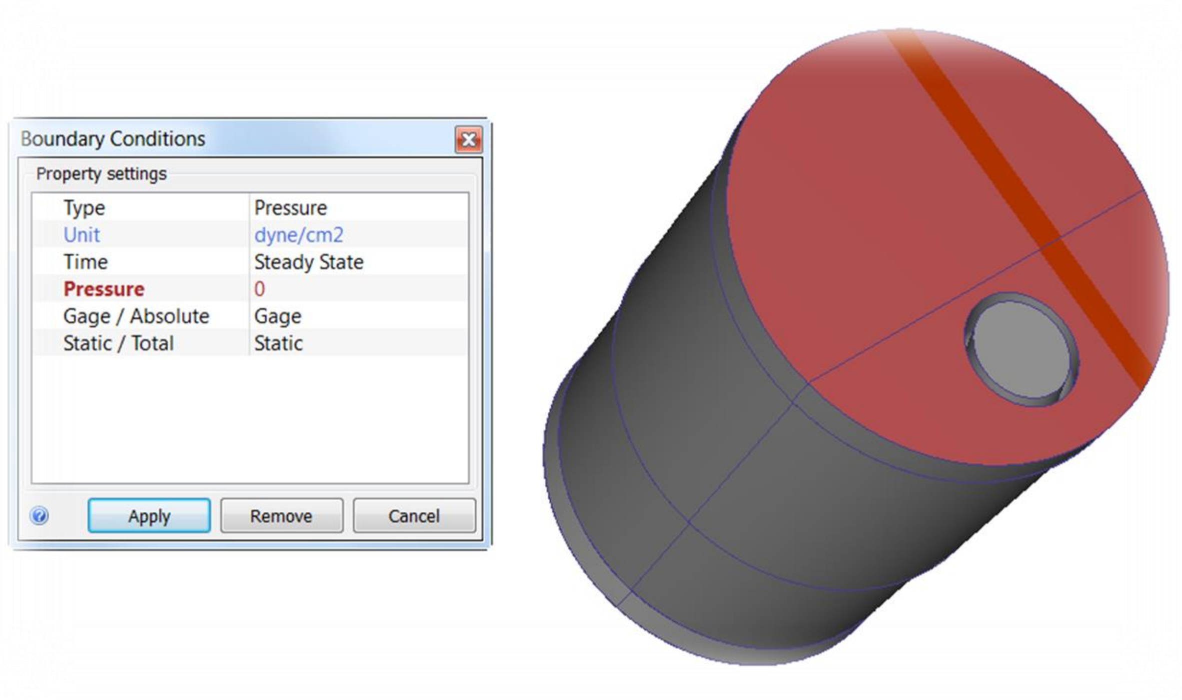


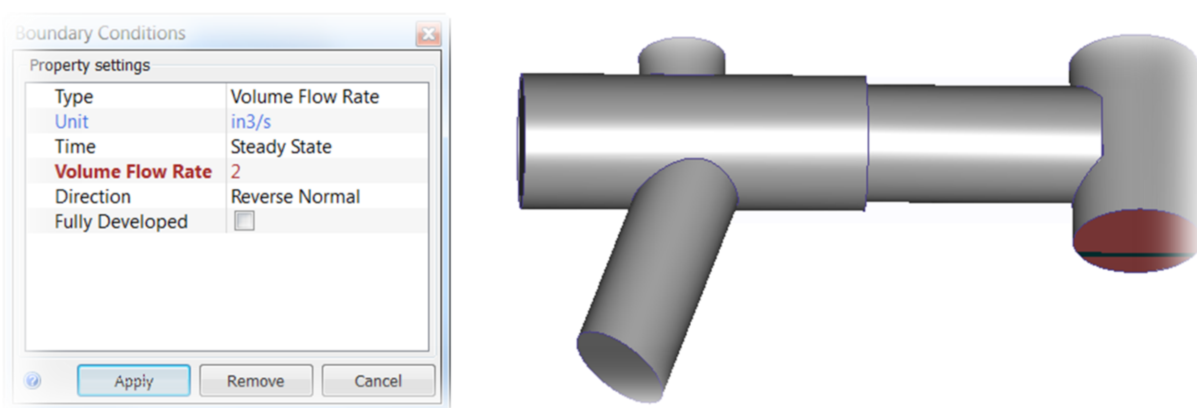
Apply Loads

Step 5: Choose the top surfaces of the fluid inside the beer cup Boundary Conditions → Type → Pressure → 0. Set Gage/Absolute to Gage.

Step 6: Choose the surface shown below in the keg faucet Boundary Conditions → Type → Volume Flow Rate → 2 in^3/s.

Why: Much like with heat transfer, we have to have 2 boundary conditions applied in order to tell the fluid "where to go." In this case, we are applying a volume flow rate in the keg and a pressure of 0 in the cup to give that beer a place to move.



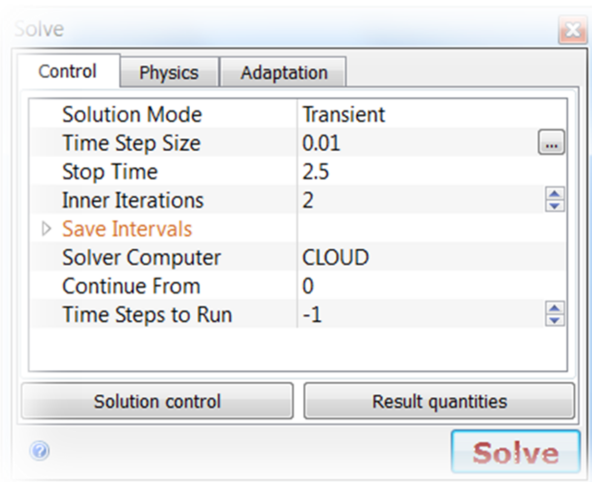


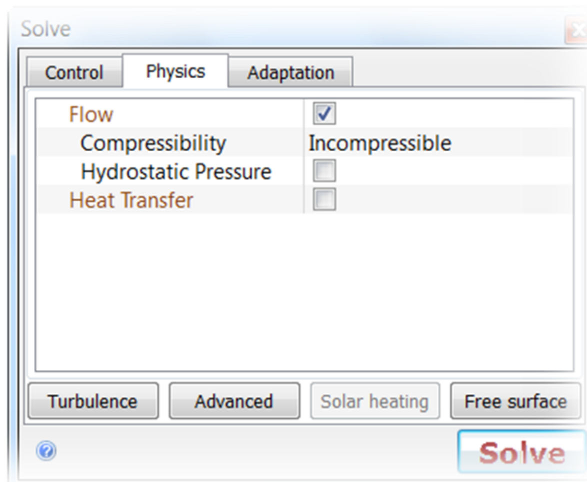
Solve the Analysis

Step 7: Press Solve → Physics tab. Check the box next to Flow.

Step 8: Press Control Tab → Solution Mode → Transient. Define settings as shown in the image below and press solve.

Why: For open channel flow you must tell the program that the fluid surface is allowed to "move." In addition, to guide the flow you have to define gravity direction as an additional load.





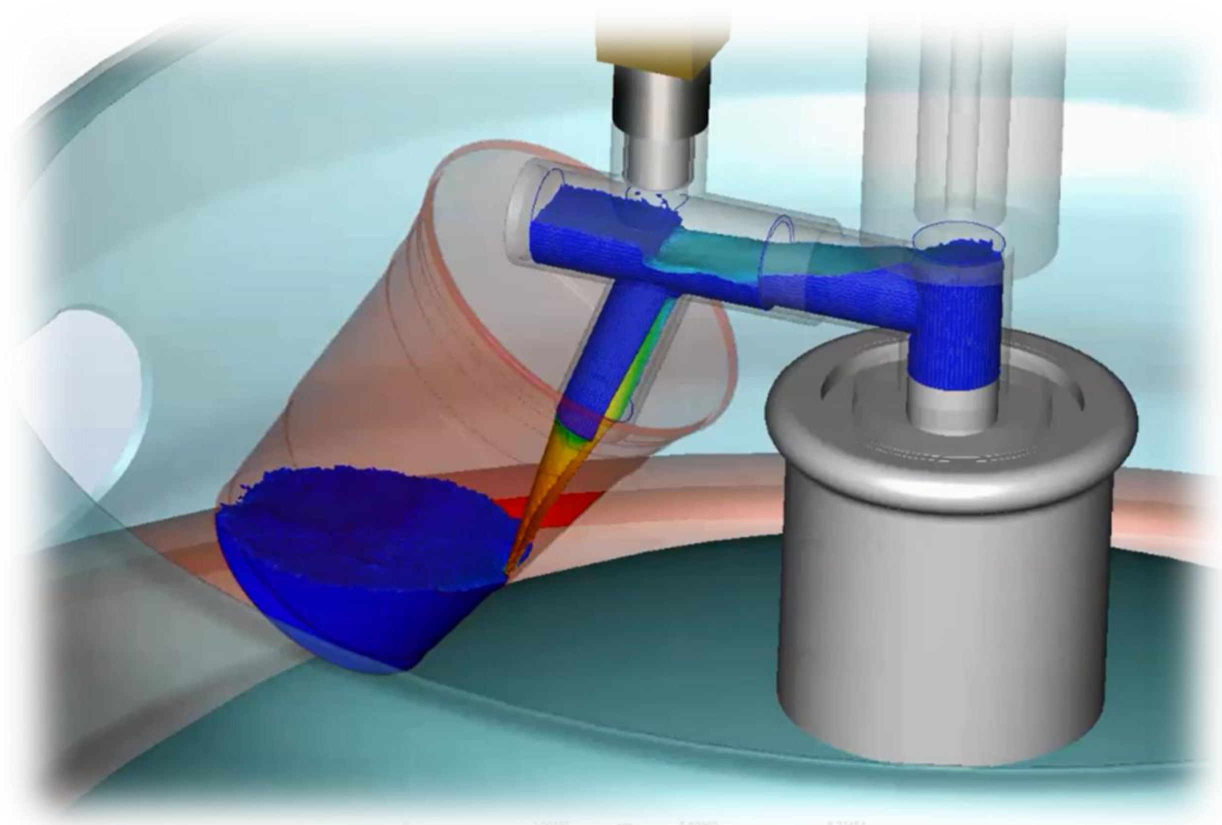
Review Results

Step 9: Press Results Tasks → Iso Volumes.

Step 10: Press Enable Visualization.

Step 11: Press Image → Animation →Select all → Animate → Frame interval = 100.

Why: To view your fluid move, it is best to use iso volumes and to animate results to see the flow. It is also important to turn on part transparency to understand how the fluid is interacting with the rest of the assembly.



Additional Autodesk Simulation Solutions to this Problem

- Just Autodesk CFD at the moment!