IEI Innova Engineering
1 Park Plaza Suite 980 Irvine, California 92614

Case Study- Importing "As-Molded" Plastic Part Conditions into CAE tools

CONTENTS

EXECUTIVE SUMMARY

Increasingly, FEA analysts are frustrated with poor correlation between the predicted results of injection molded parts subjected to FEA, and the actual physical behavior.

This case study examines methods used by Innova Engineering, an engineering firm dedicated to the analysis and design of thermoplastics parts and products.

The best way to insure accurate FEA results on molded parts is to capture the effects of the injection molding process on the structural material properties. Molding conditions can have significant effects on the structural performance of a typical thermoplastic component, and these effects can be difficult to capture and model effectively.

Most FEA analysts, even those that use non-linear codes, utilize isotropic material properties for their FE models. This case study will show that this approach can introduce significant errors into the results for certain materials, and describe a method to use actual "as molded" material properties in an analysis of a plastic part.

Approach

The approach which we will use is centered around the importance of capturing the as-molded part conditions, and importing these properties into a typical FEA solver for loads analysis.

To do this, we will employ two commercially available FE codes, for molding simulation we will use Moldex3D, for the quasi-static loads simulation we will use MSC.MARC. Both are robust non linear codes proven in commercial analysis of thermoplastic parts.

Part description

Figure 1 shows a sample part created to illustrate the case study:

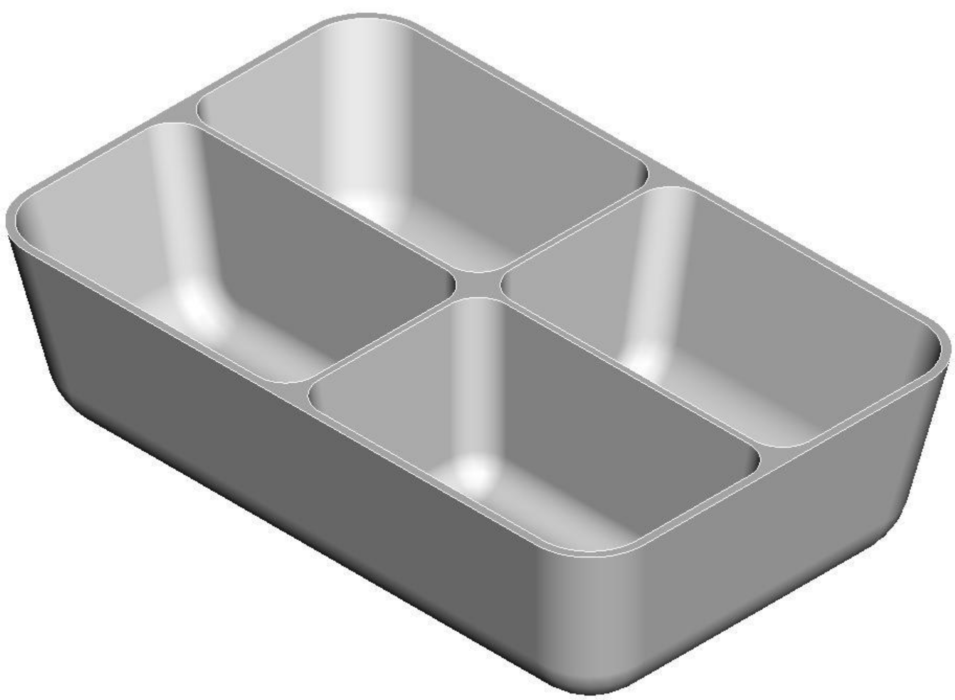

Figure 1.) Solid model of case study molded part.

Our sample part measures 10” x 6” x 3” deep. The perimeter walls are uniformly .13 thick, with the ribs measuring .10 thick. The material is 30% glass filled polypropylene.

As Molded Conditions

There are a number of factors that are of interest to the analyst which occur during the molding of a typical thermoplastic component, and these factors can and do influence the material priorities and field behavior of the parts

1.) Deformed mesh. The tool designer is aware that plastic materials shrink, and attempts to provide a correction factor (shrink factor) into the part geometry to compensate for the inevitable shrink. The exact amount of shrink to apply can sometimes be difficult to determine, as this factor is not only geometry (part) dependent, but also depends on the gate type and location as well. Running a mold flow analysis with proper PVT material curves will provide a accurate measure of the actual material shrink. The deformed mesh can then be exported to provide an exact representation of the as molded wall thicknesses and specific feature dimensions, which is much more accurate than just applying generalized scale factor in the flow and transverse flow direction.

2.) Thermal strains. Differential cooling of the part can lead to thermally induced strains that contribute to distortions in the final geometry.

3.) Residual stress. Two factors can lead to residual stresses in the molded part, thermally induced stresses, and flow induced stresses. Both result in build up of stress in the finished part. These stresses should be considered in any downstream structural analysis of the part performance, although the values are usually quite low. They can and do create conditions where temperature cycling, sterilization, and long term exposure create dimensional variances in the part.

4.) Weld lines. Intersecting melt fronts create weld and meld lines in any part, these areas have different structural material proprieties than homogenous material, and if these weld lines occur in areas that carry load, we have to accommodate these reduced properties in any load bearing calculations.

5.) Flow line orientation. Particularly for filled materials, this is one of the most critical of all criteria, and the one most often compromised. The mechanical properties of thermoplastics is highly dependent on the flow orientation of the plastic as it enters the mold cavity, and also dependent on the part shape to determine flow line orientation. For glass filled materials, this means the materials are highly anisotropic, and using generalized mechanical properties for FEA is a dangerous assumption. This case study shall focus on this aspect of the as-molded part to illustrate the issues at hand. Figure 2 illustrates the high variability for a glass filled material- the top shows a actual section of a molded part, the bottom shows the predicted results from the simulation.

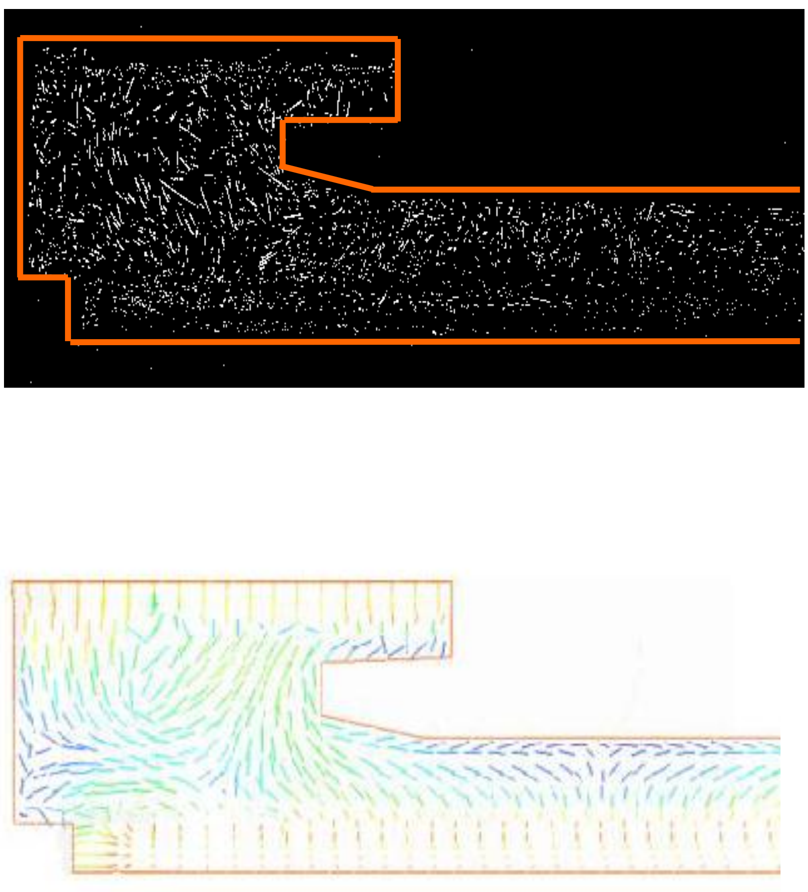

Figure 2.) Flow orientation of glass fibers

Deep Dive

Mold Flow Simulation

We are considering a sample part as shown in Figure 1 molded with a 30% glass filled polypropylene polymer. We have selected a gate location to illustrate the appearance of weld lines, and to show the flow behavior of the plastic as it enters the cavity.

We have set up a simple mold flow simulation using approximately 500,000 full 3D elements. It is critical to use full fidelity 3D elements for this type of simulation, the typical CAD plug in type of solver using a mid-plane model and 2D shell elements will not capture the flow line behavior properly. The simulation included filling, packing, cooling, and warp loadcases.

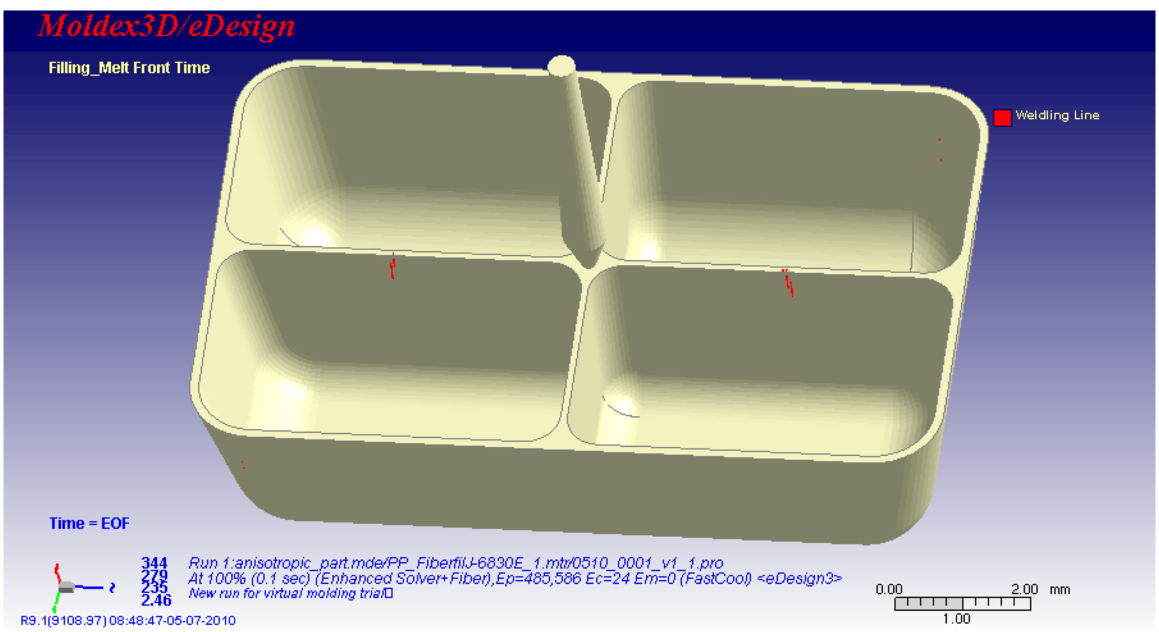

Figure 3.) Mold filling simulation.

Clicking on the animation in figure 4 shows the filling process, and highlights the weld lines to the left and right of the gate location.

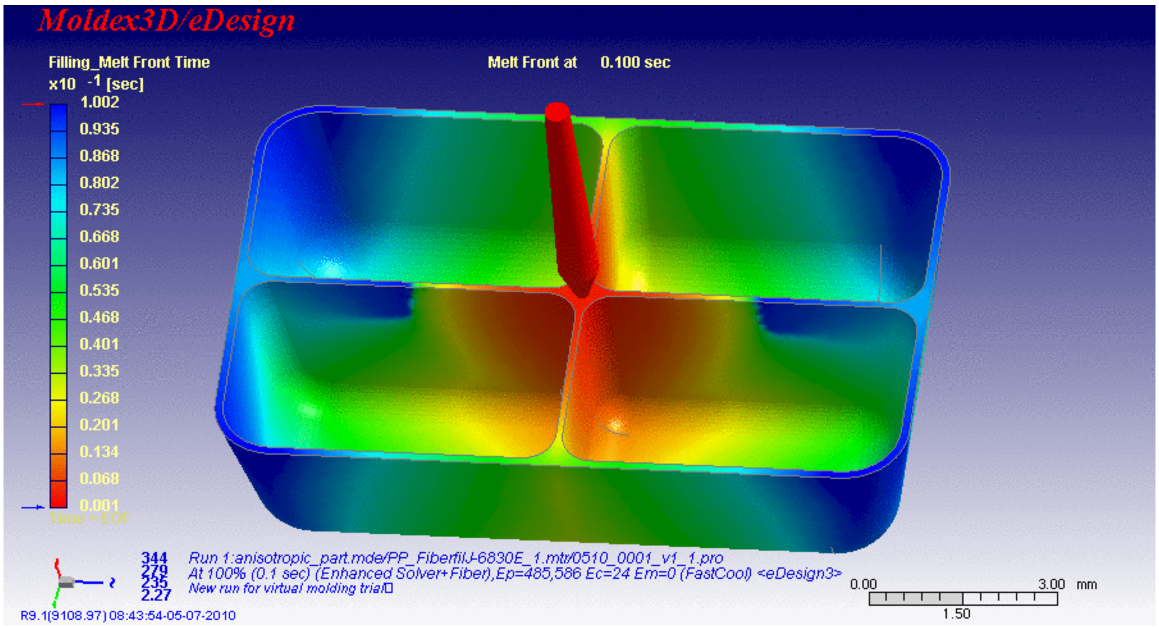

Figure 4.) Filling animation

Filling_Melt Front Time.gif

After our filling simulation is complete, we can open our post processor and view the fiber origination of the 30% glass filled material. Figure 5 shows the entire part with the directional ordination of the fibers highlighted. Very strong axial orientation is seen at the sides, and as the plastic “turns the corner” in the outside radii, the directionality of the fibers changes as a result of the flow dependency and geometry influence.

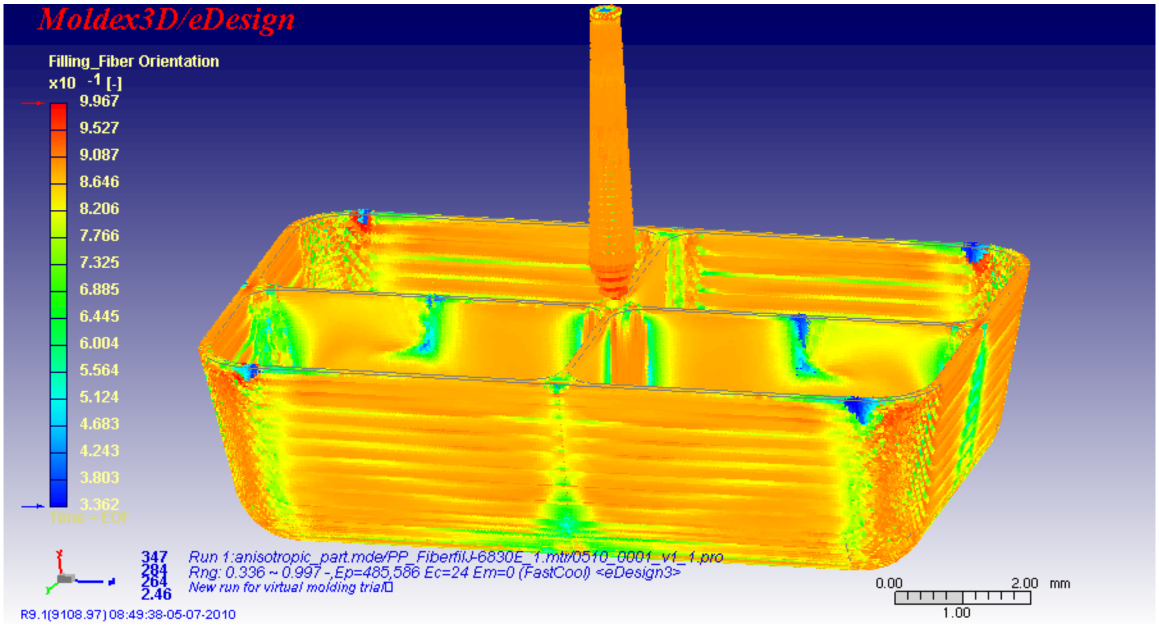

Figure 5.) Visualization of fiber orientation

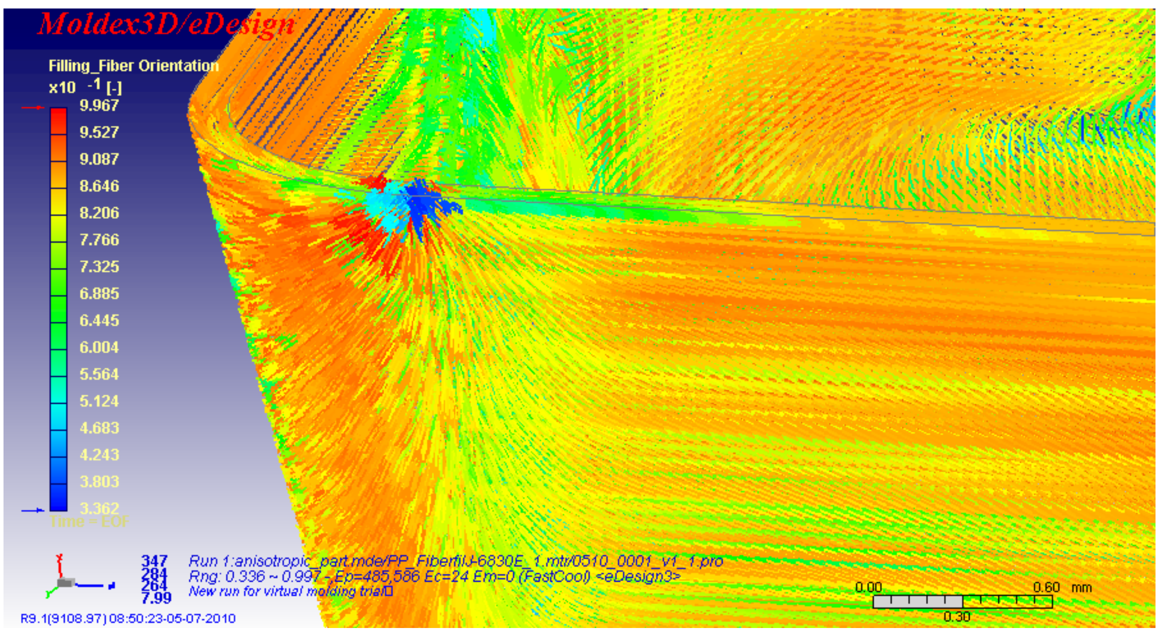

Figure 6.) Fiber orientation- close up

What we are seeing in figures 5 and 6 are fibers represented by directional arrows, or vectors, corresponding to the flow induced orientation. The darker the color, the more directional orientation, the blue represents randomized fibers.

Much can be learned just from the visualization of the directionality. Designers can consider the areas of the part that will see load, and they can insure the load is as close to parallel to the fiber direction as possible, as this is where the material has the greatest strength. The sample can be of course considered for the weld liens, we want to be sure the weld lines are not located in a load path if possible.

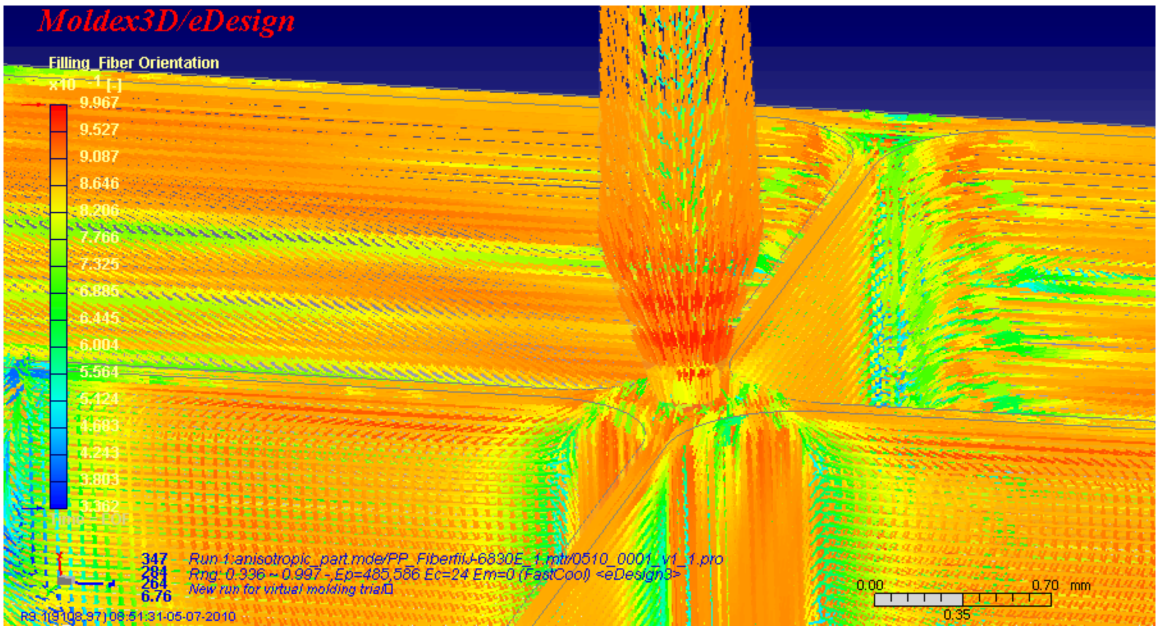

Figure 7.) Fiber orientation- runner to gate location

Beyond these visualization techniques, we must understand the structural capacity of the material in the as molded condition. The flow lines in this example are so nicely orientated that we could consider the properties to be orthotropic, and this would be the case for the majority of the part,. But we will use anisotropic properties so as to capture the randomized areas that do not exhibit cleanly orthotropic properties. Examples of these areas may be seen in figures 5 and 6 as noted ion blue.

As a reference, isotropic properties are what is normally used on FEA of plastics, this considers the mechanical properties to be the same in all directions of orientation. Orthotropic considers the properties to be different in X, Y, and Z, and anisotropy considers the mechanical properties to be different in all directions, the most comprehensive material property model available. This is the model we shall use.

Now that we have identified the flow line orientation through a comprehensive 3D mold flow simulation, we can now take steps to export these properties to our downstream FEA solver, capturing the as-molded condition as anisotropic material properties, and mapping these material orientations to the new FE mesh.

Structural FEA Simulation

To set the stage for a robust comparison, we intend to create two identical FE part models. The first will use the most commonly used method by plastics analysts, which is to use isotropic material properties. This can be published data, as is often the case, and this usually means Young's' modulus and Poissons ratio if a linear elastic loadcase is anticipated. Sometime, the FE analyst is keenly aware of the pitfalls of using linear elastic analysis for plastics, and will instead perform physical testing to develop elastic-plastic stress strain curves. In either case, the result is usually the assumption that the material behavior is isotropic.

The second model we will run will use the exported material properties as determined in the previous mold flow study, and will take full advantage of the flow orientation of the glass fibers. We will load the parts identically, and examine the results.

To start, we will create a mesh in MARC which will be imported into the mold flow solver for mesh mapping. Once imported, the fiber orientation results are mapped to the new mesh. This mesh is now brought back into the MARC non-linear solver, and standard boundary conditions are applied. Our problem in both cases is to be displacement controlled, e.g. we will apply a displacement of .12/inches to the center of both parts, and solve for the maximum stress in each part.

Figure 8 shows the mesh of the as-molded part. We can clearly see the mapping has taken place, each color gradient represent some different anisotropic material value depending on fiber orientation. There are some 1600+ different mechanical property values differing as a result of the flow induced orientation.

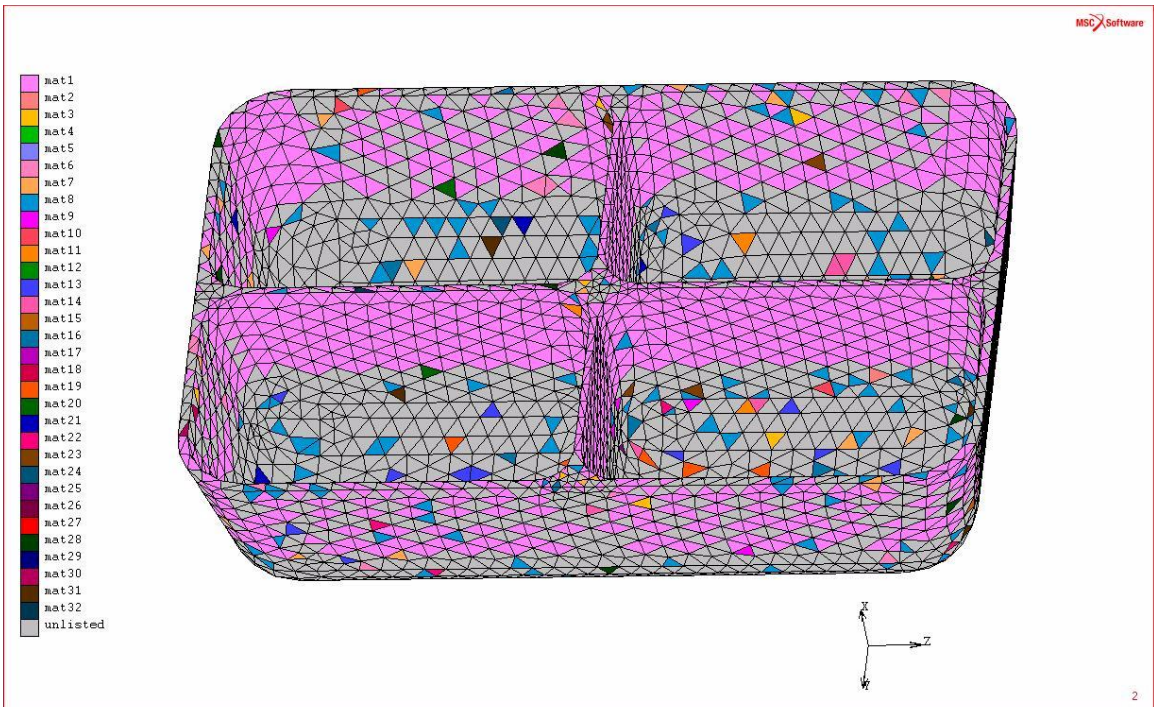

Figure 8.) Fiber orientation- mesh mapping

Results

Boundary conditions were established on the short side segments, and the load applied in the center as described earlier. First, we will look at the typical isotropic material model results.

Figure 9 shows the peak Von Mises stress when the part is loaded without consideration for the as molded material condition. In other words, we do not take into account the directionality of the fiber orientation. We show a peak stress value of 70 Mpa. This particular material has a yield value of 56 Mpa, so this part is well on it's way to failure, as this analysis indicates unacceptable part performance and will likely necessitate a redesign

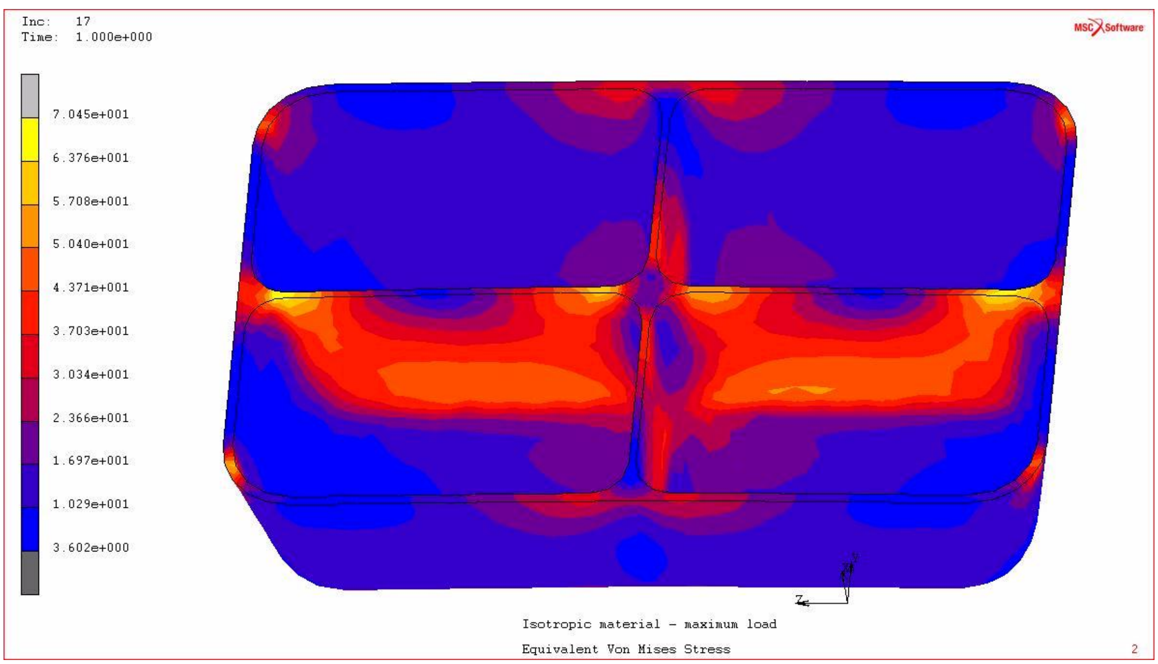

Figure 9.) Isotropic Material Model

Now, let's rerun the job with all conditions identical, except this time we will consider the as molded material conditions, which is to say we will take into account the directionality of the glass fibers. Figure 10 shows us the results of the same loadcase using the anisotropic material model.

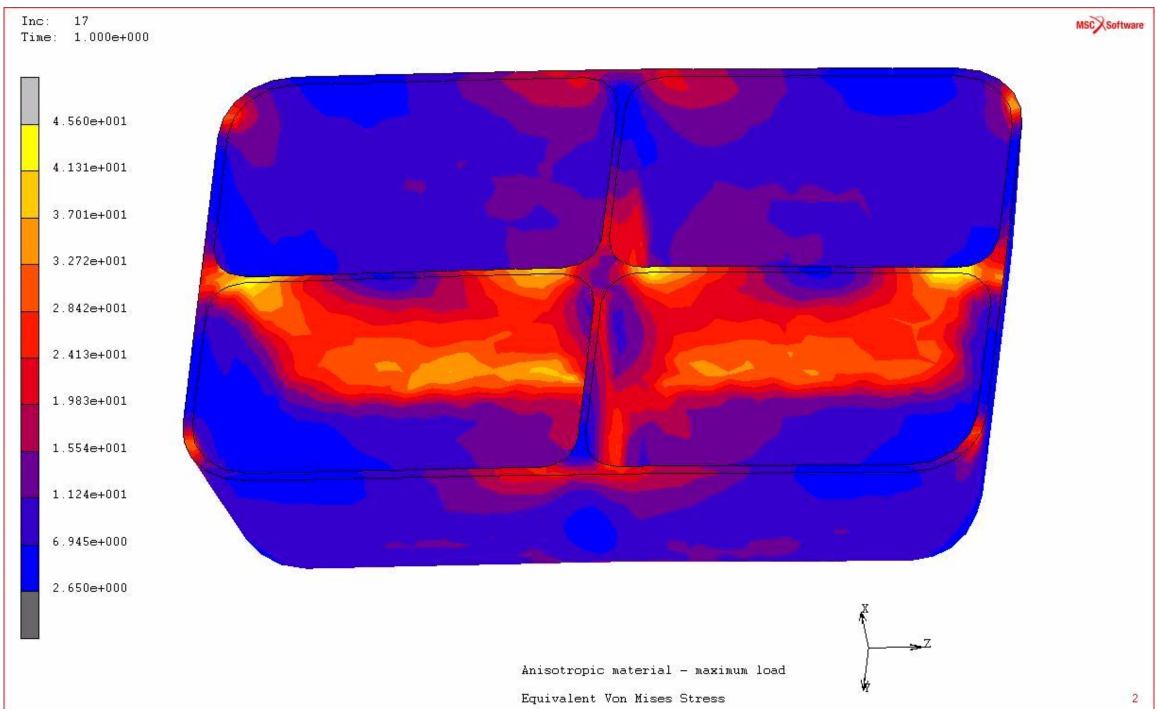

Figure 10.) As-Molded (Anisotropic) Material Model

The difference in stress magnitude is quite dramatic. We now see a peak Von Mises stress of 45 Mpa, well under yield, and gives us a very different version of the structural performance of the part with this material.

Conclusions

In the case of this 30% glass filled material, using the standard isotropic material properties yields highly inaccurate results.

The predicted stress magnitudes differ by 55%, a substantial error. It is noteworthy to mention that this loadcase was displacement based, meaning the same fixed deflection was introduced to both parts as described earlier. If the analysis was load based, meaning the same force was applied to each part and the resulting deflection was allowed to vary, the effect would be more than 50% predicted deflection for the isotropic part instance.

Most materials have some flow directionality when molded, especially filled materials. If an accurate assessment of as molded behavior is expected, it is important to capture the flow induced orientation of the material.