Advanced simulation of 3D glass bottle forming with Abaqus/CAE, Abaqus/Standard and Abaqus/Explicit

Gerard de Leede, Rik Koch

Glass Service Improve BV

Vincent Bouwman, Gertjan Kloosterman

Abaqus Benelux BV

Abstract: Production of glass bottles requires blowing of the glass after entrance of a gob of molten glass in the blank mould. The final shape of the bottle is highly dependent on the viscosity of the glass, the blow-pressure and the temperature distribution in the glass and the mould and simulation of this complicated process enables optimization of the process conditions. During simulation of blowing of the glass, the mesh has to be adapted due to the extreme deformations of the mesh. Using the existing ALE-technique for this kind of applications requires a lot of user-intervention and trial-and-error to create a mesh that suits both the initial and final topology of the glass. To reduce the user-time and to be able to run this kind of analyses automatically based on an arbitrary base-geometry, a completely automated remeshing/rezoning procedure is set-up. In this procedure the A/Explicit analysis is divided in a number of sub-analyses after each of which a new (3D) geometry of the glass is created based on the deformed mesh (using ABAQUS/CAE). Using a map-routine the solution from the previous analysis is mapped on the new mesh such that continuation of results is ensured. Using the automated remeshing capability, simulations of the glass bottle forming process have successfully been performed, enabling for example optimization of process settings. Due to the generic set-up of the remeshing procedure it can easily be used for other simulations that require adaptive meshing as well.

Keywords: Coupled Analysis, Glass Forming, Remeshing, Scripting, Viscoelasticity, Visualization.

1. Introduction

Current container glass manufacturing is mainly driven by experience and craftsmanship rather than scientific knowledge. There is a lack of understanding of the complex glass forming process as well as of the application of (scientific) knowledge and tools. Troubleshooting at the design and production stages is based on experience and new container designs are introduced by trial and error. The trial & error stage is usually repeated several times over and can last several months. These stages are very expensive, as tests can only be carried out in a live production environment. Market demand however no longer allows for lengthy trial & error periods.

The glass industry is now forced to combine ever more physically complex designs with extremely tight production and delivery times. In order to meet these delivery times, insight into the process prior to the actual production phase is required. So a general approach is required for setting up analyses for glass forming processes.

1.1 The blow-blow process

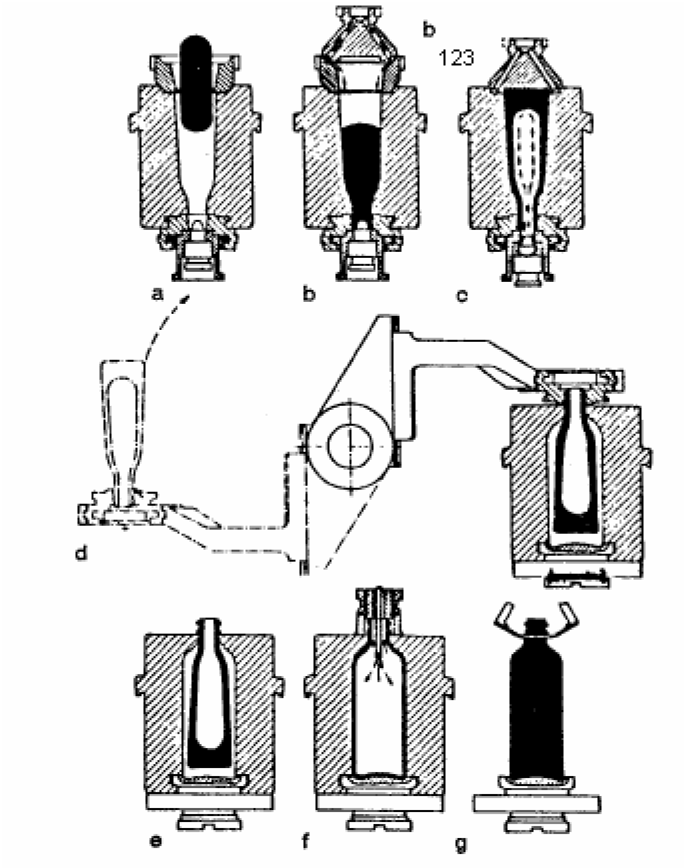

Figure 1. The blow-blow process.

The glass forming process that is studied in this paper is the blow-blow process as depicted in Figure 1. The total process consists of several stages among which are two forming stages which both incorporate a blowing of the glass. A gob of glass enters the first mold in step ‘a’. The gob moves down in the tools and the neck is attached by a settle blow. After the proper creation of the neck, in step ‘c’ the first forming blow is performed, which is called the counterblow. This step induces a severe deformation of the glass. The product after the initial blow is called the parison. The parison is rotated in step ‘d’ into the final mold, held for some time in step ‘e’ to let it sag, and then in step ‘f’ it undergoes the final blow to attain the desired shape. The deformation in this step is less severe than that in the counterblow, but still the wall thickness can change significantly.

1.2 Organization

After this introduction, a short discussion on different remeshing strategies is discussed in Section 2. From this it can be seen that due to the radical geometry changes a full remeshing strategy is required. How to perform this remeshing is discussed in Section 3. Next in Section 4 an example is presented. Finally in Section 5, some conclusions are drawn.

2. Remeshing

Glass forming has not been done that often with a Finite Element Approach (FEM), since the material behavior lies somewhere between a fluid, which is the domain of Computational Fluid Dynamics (CFD) and a solid, which is the domain of FEM. The reason the glass industry has kept with the trial & error approach for so long is that there is no clear direct application of either technology to successfully simulate the glass forming problem.

The difficulty lies in the material behavior of the glass. It is feasible to do some forming of the glass as long as the deformation that occurs during the forming is not too drastic. Hence, a final blow of a parison to a bottle might be possible, making use of an adaptive remeshing technique. The counterblow from a gob to an initial shape however, incurs extreme deformation of the glass material. As a result the underlying mesh of the FEM solver will also distort severely, so much so that the original mesh topology can no longer represent the final shape.

2.1 Remeshing techniques

During the forming of the glass product, the deformation of the material is severe. This is because at high temperatures the glass behaves as a viscous fluid. Moreover, the geometry change that the gob needs to undergo is huge. The surface area will increase by a factor of 2 whereas the volume will remain the same. You can see this difference from studying phase 2 of the blow-blow process with phase 5, which are the respective start and end point. Thus, it is difficult for a given mesh topology to represent the two extremely differing geometries. A topology specifies how things are connected together logically: it does not consider shape. So if you take a sphere, and flatten it out to an ellipsoid they will have the same topology, but a different geometry. If however you take a sphere and a donut, these two objects also have a different topology: the sphere has no hole.

If we now would compare two meshes on two different geometries, and if you would be able to make a labeling on the nodes and elements, such that if element A shares a face with element B in both the first as well the second mesh for arbitrary A & B, then the two meshes have the same topology.

Based on this consideration, we can consider two types of remeshing algorithms: remeshing algorithms which retain the original mesh topology and those that do not. Abaqus contains an implementation of the first technique in the form of ALE within the solvers. ALE stands for Arbitrary Lagrangian Eulerian. In the ALE algorithm, the mesh initially follows the motion of the material points. Then at a certain point in time, the mesh motion is decoupled from the material motion to optimize the element shape. The solution is then mapped using an advection scheme, and the analysis is continued. The ALE algorithm thus tries to optimize the original mesh topology onto the new geometry which arose due to deformation. However, as was already discussed before: In glass production the geometry changes in an extreme sense. This in turn means that the original mesh topology will not fit onto the new geometry without having distorted elements. It is for a finite element solver not possible to continue the calculation with distorted elements, and the analysis will thus end prematurely.

From this we can easily conclude that in order to analyze the glass forming problem and other problems that undergo such significant deformations, a full remeshing of the computation domain is to be performed. There is no such algorithm available by default in Abaqus, thus Abaqus Benelux BV implemented a first order technique. To be usable by the typical design engineer working in the glass market, it also needs to be robust and fully automated.

Since it needs to be possible to mesh arbitrary geometries during the remeshing step, the model is set-up using modified formulation quadratic tetrahedral elements with both displacement and thermal degrees of freedom. So that the glass will not exhibit volumetric locking, nor should it have problems with contact. Both temperature and displacement are required, since the viscosity of the glass changes strongly with temperature, and the temperature changes strongly if the glass touches the mold anywhere.

2.2 Remeshing simulation strategy

The counterblow and finalblow analyses will be set up using the following strategy:

1. A number of subjobs are chosen for the total analysis, let us say *n*, this will divide the total analysis history into *n* distinct jobs, which taken together form the complete job.
2. After each subjob, the quality of the mesh is checked. On the quadratic tetrahedral elements the face corner angle is checked as well as line angle on the midside edge nodes.
3. If the element quality is still ok, the next subjob is a restart of the current subjob.
4. If the element quality is poor, the analysis needs to be remeshed. The next subjob needs to have its solution then mapped as well.
5. Once all the subjobs are finished, the results need to be visualized in an easy way, so the results will need to be combined.

As can be seen there are already several ingredients here that need to be developed without even considering the remeshing yet:

- A driver script is required that runs the subjobs in sequence and decides what to do.
- An element check routine is necessary
- A remeshing routine is necessary
- A solution mapping scheme is necessary, since there is no equivalent *MAP SOLUTION capability in Abaqus/Explicit.
- A script which performs similar the restartJoin script in Abaqus is necessary, but this time based on remeshed instances.

All this has been implemented, and the discussion continues with how the remeshing itself is performed.

3. Remeshing using Abaqus/CAE

In order not to have to write a complete mesher and having to do the model set-up again in a different package, as much as possible of the technology in CAE that was available is re-used. This means that the model is kept, and all that is done is to replace the instance in the original model with the remeshed one. As soon as that is done, the new input file can be written out. In order to make the exchange of the old part with the new part as smooth as possible, the CAE model is required to be setup using only sets and surfaces. Hence, the user should not pick anything in the viewport, but instead define all sets and surfaces on the part level. Everything in CAE is then referred to by name, and exchanging one part-instance with another having exactly the same sets and surfaces is then seamless.

The only issue to be solved is how to remesh the instance in the analysis. Moreover, the user should not have to define the sets and surfaces after every remesh increment, since this would be cumbersome; rather the sets and surfaces ought to be recreated from the deformed mesh piece. The remeshing strategy is thus as follows:

1. From the outside of the mesh a new geometry is created.
2. Using the original and the deformed mesh, along with the initial geometry of the instance with the mesh on it, all the topology regions are re-identified.
3. Using this topology map, the sets and surfaces can be reconstructed.
4. Finally the instance is replaced in the model, all boundary conditions are adapted to account for the time passed, amplitudes are fixed up, and the analysis is written out.

The total bookkeeping to get all this done is very complex, but quite efficiently automated. An application example is presented next.

4. Example of simulating a blow-blow process

The automated remeshing capability as explained in the previous section has successfully been used in several bottle forming analyses. In this section an example of the Italian company ‘Bormioli Luigi’, known for it’s manufacturing of glass containers for the perfumery and cosmetics industry, is presented.

Pictures of the final shape of the bottle of consideration are shown in Figure 2. From Figure 2, the complexity of the shape of the bottle and the variations in glass wall thickness are clearly visible.

Figure 2. Pictures of final bottle

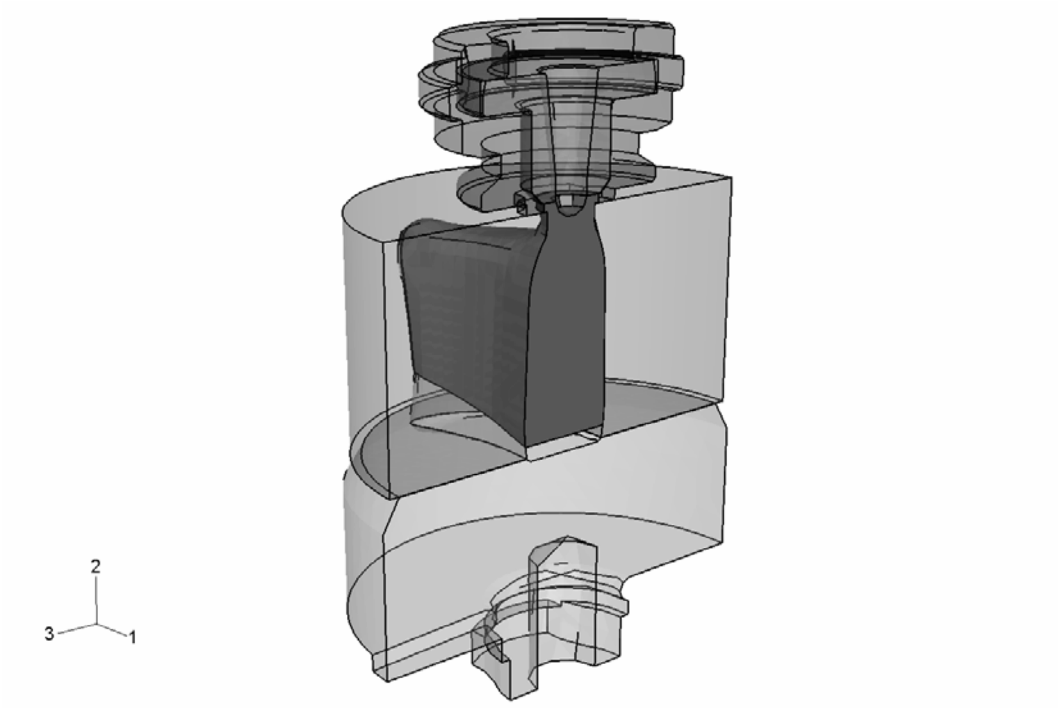

Figure 3. Shape of the gob in cavity of the tools

For a typical bottle, half of the assembly of the tools and the gob of glass prior to blowing is depicted in Figure 3. The translucent parts are the tools from which the shape of the cavity that has to be filled with glass becomes clear. The small spherical face on top of the gob is the face on which the blow-pressure is applied and because of the ‘sticking’ behavior of hot glass, contact between glass and tools is modeled as a no-separation contact with rough friction properties. Due to the complex geometry and to allow automated remeshing, the gob is meshed with quadratic tet-elements C3D10MT.

Glass at high temperatures is quite a difficult material to use in finite element analyses. The reason for that is quite simple: It behaves as a fluid, with a strong temperature dependent viscosity. The most direct approach to model glass in Abaqus is to use an equation of state material model (EOS). However, using the glass material parameters in an equation of state material model in Abaqus/Explicit will give very small stable time increments, leading to long computation times. In order to reduce the calculation time, we have regularized the glass material model to a specific elasto-plastic material model such that it still gives accurate results, but also has a much larger stable time increment for the same element size. The actual details of the material model are out of the scope of this paper.

The simulation of the complete blow-blow process as described in paragraph 1.1 is simplified and consists in this example of the following stages:

1. Transient heat transfer analysis (A/Standard) from the moment the gob arrives in the mould till the start of the counterblow. This will result in the temperature distribution necessary for the next stage.
2. Coupled stress-displacement analysis (A/Explicit) of the counterblow. This will result in the shape of the parison and the temperature distribution during the process.
3. Cyclic transient heat transfer analyses (A/Standard) of the complete process to obtain the ‘steady-state’ temperature distribution in the tools and the glass. Starting with estimated initial temperatures, every subsequent analysis starts with the temperatures from the previous analysis. The complete cycle is assumed to be converged when the temperature distribution in all points during the analysis in two subsequent analyses differs not more than tolerance (in this example 1º).
4. Quasi-static analysis (A/Standard) of the ‘elongation’ of the parison. This will result in the shape of the glass and the temperature distribution necessary for the last stage.
5. Coupled stress-displacement analysis (A/Explicit) of the finalblow. This will result in the final shape of the bottle and the temperature distribution during the process.

The automated remeshing technique as explained in previous sections is used in the stages 2 and 5, the actual blow-simulations. Figure 4 illustrates one of the remesh operations during the counterblow analysis. The left picture shows the mesh of the gob at a certain moment, while the middle picture shows the same mesh after 4 [ms] of blowing with a pressure of 2 [bar]. Clearly visible is that some elements become too distorted and remeshing is required. The right picture shows the ‘new’ mesh of the gob after the remesh operation.

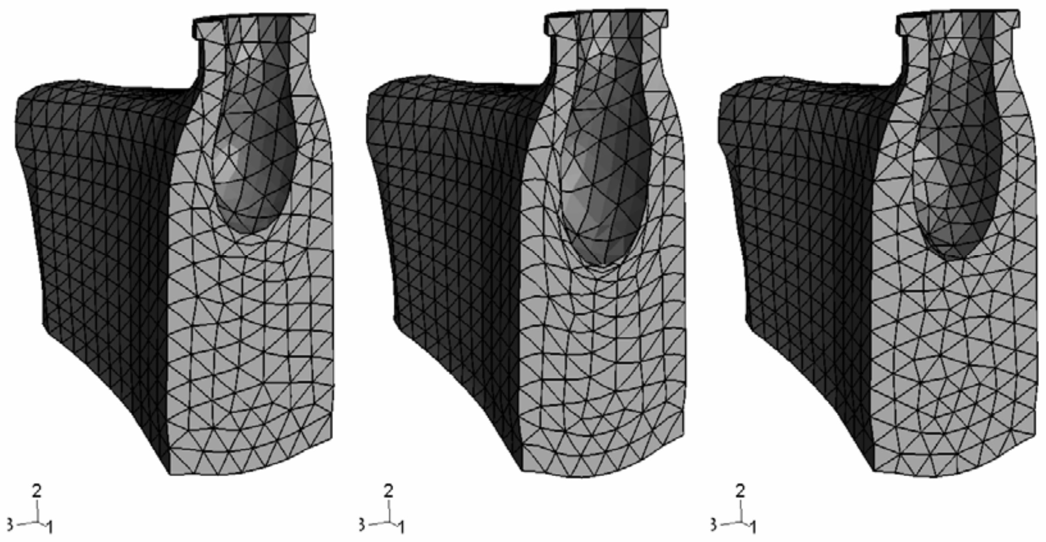

Figure 4. Mesh of the gob prior and after remeshing operation.

From Figure 4 it becomes obvious that without remeshing the analysis will abort quickly due to mesh distortion, even ignoring the effect that prior to the abort the stable time increment will drop drastically because of squeezing of elements. Including remeshing however allows the analysis to proceed easily.

With the specified tolerance on element quality, the complete counterblow analysis finished with 9 remesh-steps after which the cyclic thermal analyses start. The temperature distribution on the final shape of the parison and the tools is shown in Figure 5.

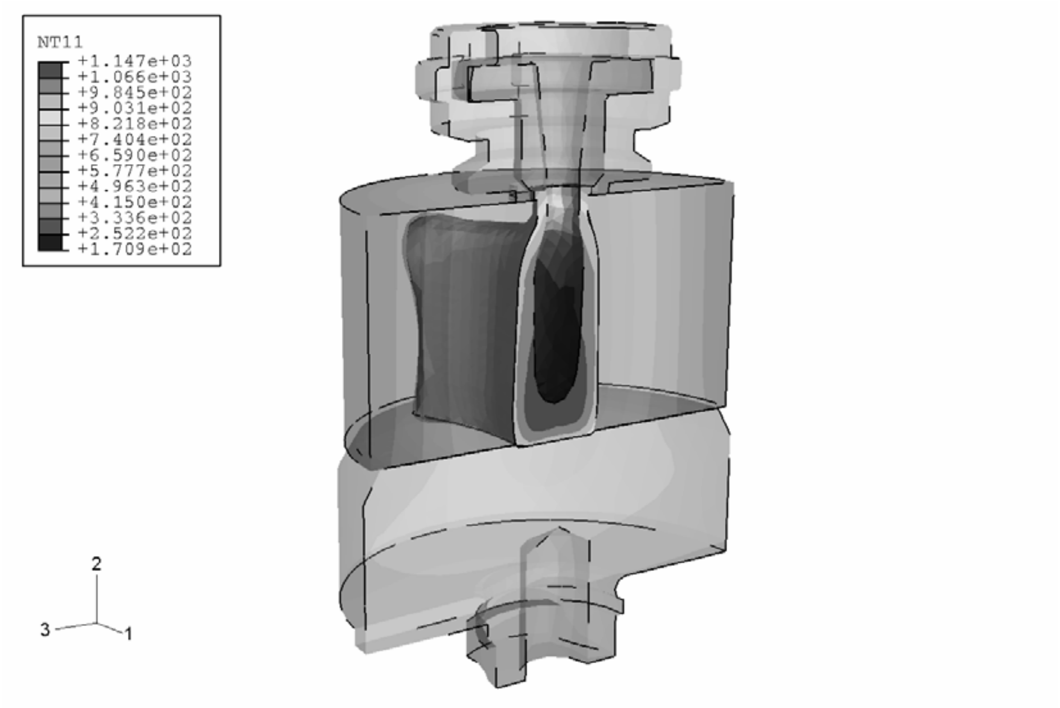

Figure 5. Temperature distribution on final shape of parison and tools.

The temperature distribution on the parison after the counterblow is validated with infra-red measurements on the real product. In Figure 6 these measured temperatures are given together with the results of the simulations. The continuous lines in Figure 6 indicate simulated temperatures after opening of the mould, where the ‘path distance’ as given on the horizontal axis is measured from the ‘neck’ to the bottom of the parison. Due to heat-redistribution from the inside of the parison, the temperature at the outside of the parison increases after opening of the mould, which is indicated by the two lines which are obtained at the moment of opening of the mould and some time afterwards. The markers in Figure 6 indicate the measured temperatures which are all in between the two curves and show the same trend over the length of the parison.

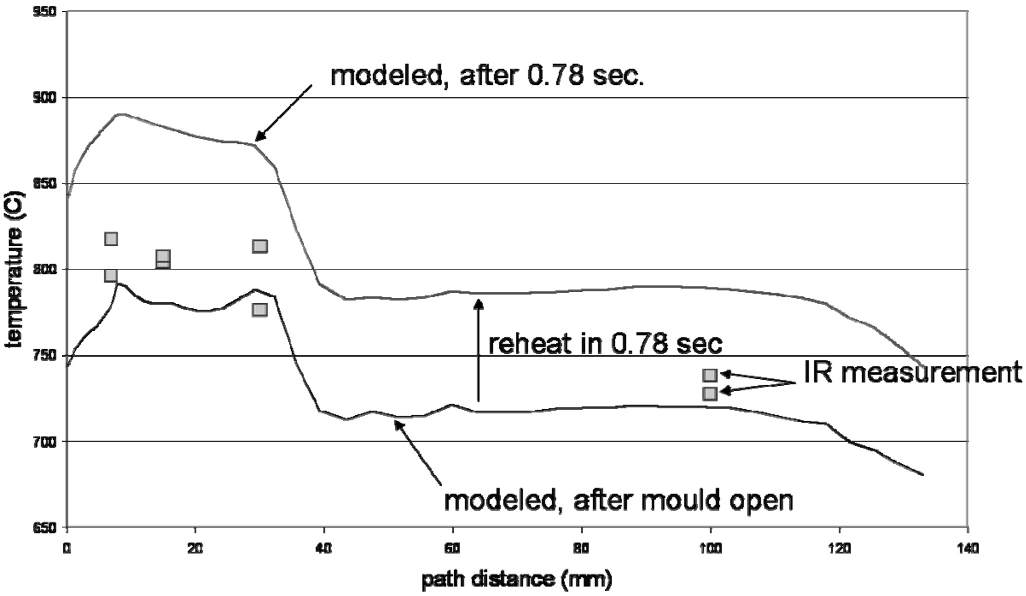

Figure 6. Simulated and measured temperature distribution on the parison after opening of the mould.

The final shape of the bottle is obtained in the last stage of the simulation sequence, the ‘finalblow’. The cut-plot in Figure 7 shows the shape of the parison just before the start of the finalblow and the final shape of the bottle in the cavity of the final mold.

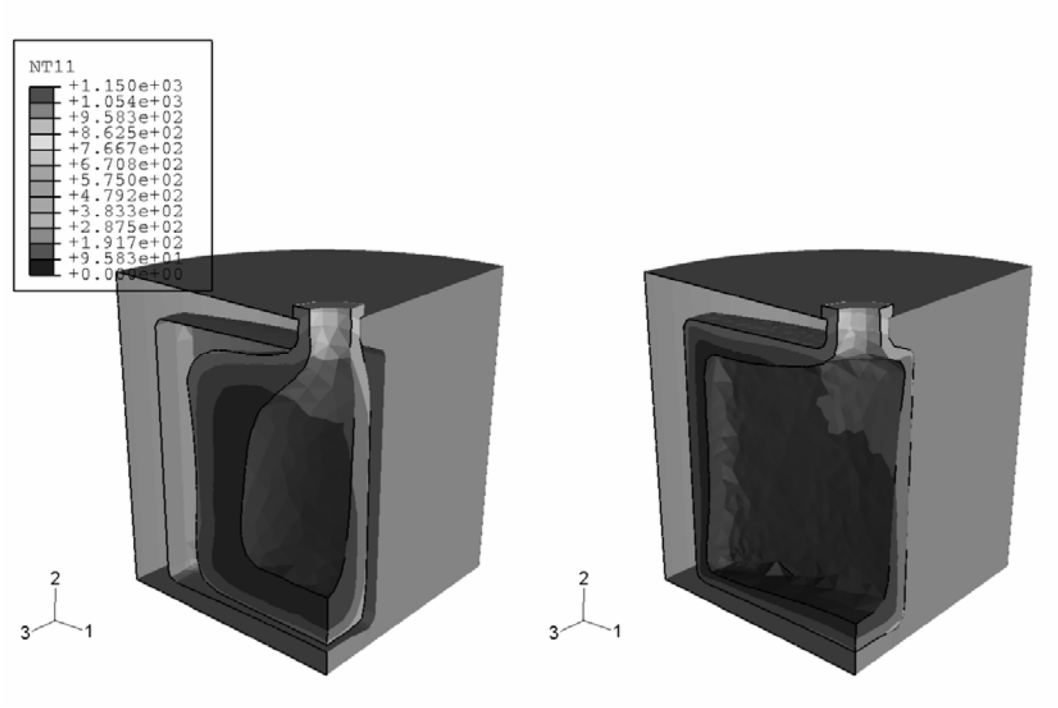

Figure 7. Shape of parison just before and after finalblow.

The need for remeshing becomes apparent again from Figure 7 although the deformation is less severe compared to the counterblow analysis. The finalblow analysis finished with 4 remesh steps and in Figure 8 finally the shape of the glass is displayed at different stages during the bottle forming process.

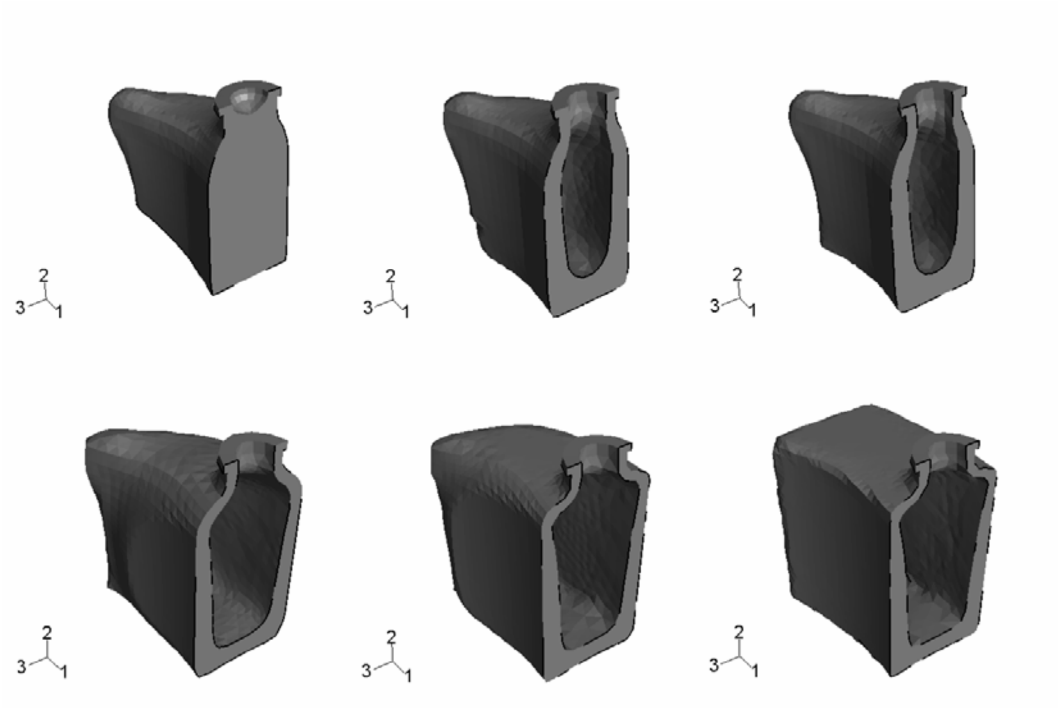

Figure 8. Shape of glass at different stages of the bottle forming process.

The complete analysis sequence took about 15 hours on an ordinary laptop using ABAQUS V6.6. Being able to perform this kind of analyses without user-intervention now allows next steps to be taken such as optimization of the glass wall thickness distribution by modifying process conditions.

5. Conclusions

From the example we can see that even though the (Lagrangian) finite element method is not really laid out for the enormous deformations that occur during glass forming, it is still possible to simulate it using advanced remeshing techniques. The remeshing technique fully exploits the capabilities of Abaqus/CAE in terms of geometry handling and meshing.

The remesh technique is developed by Abaqus Benelux BV and successfully tested on complex shaped bottles in cooperation with GS Improve. Due to the generic set-up of the remeshing scripts, which are the sole property of Abaqus Benelux BV, the usage is not limited to glass forming problems. There are very likely other bulk-forming problems which are challenging and we hope in the future to use this technique also in those applications.

6. References

1. Bird, R.B., W.E. Stewart, and E.N. Lightfoot, "Transport Phenomena", 2nd ed., John Wiley & Sons, New York, 2001

2. Kloosterman, G., and R.M.J. van Damme, "An orientation on surface reconstruction", Memorandum no 1478, University of Twente, Faculty of Mathematical Sciences, 1998.

3. Koch, H.J., "Optimization of mould design by mathematical modeling", Society of Glass Technology ESG2006, Sunderland, 2006