BUILD SIMULATION ANALYSIS

Tutorial_V5 - 14,0200,1606,1028(SP2)

Contents

Introduction

The need behind printing or build simulation:

3D Printing in metal is a challenge, specifically when printing large volume parts.

Preparing a part involves several printing iterations to assure high quality results, and this is a usually very costly process. In addition to the actual print failure, such prints may even end in damage to the equipment.

The purpose of the analysis is to reduce the number of iterations, optimize your supports design and validate that you get the expected deformations according to the allowed tolerances.

What can possibly go wrong?

Well, you add supports to the model, but being too cautious, you may place too many supports, thus waste material and printing time. This adds also to the cost of printing and even form excess strain.

On the other hand, adding a too low number supports may result in part deformation and even lead to tears or cracks in the part.

An even worse problem might be that the material may form above the powder bed and this may end up in damage to the equipment (when using a printer with a recoater or stop the build (in case of a roller).

If the deformations that result on the part are too far from original design, then of course the part is lost, the results need to be analyzed, the design needs to be changed and a new build should be carried out.

About Build Simulation

3DXpert's Build Simulation is a macro scale estimation of the process. It is based on the geometry of the model itself (its topology and weld conditions Raising). As such, it is much faster than the actual printing time and therefore practical. This method involves calibration of the machine and material.

About Calibration

The calibration process is done in order to optimize build simulation results and thus achieve a more accurate simulation.

In general, the calibration is a preliminary process in which you print a predefined calibration part with constant geometry, then remove it from the plate and make different measurements on the part. The results of these measurements are then input to the system and then used by the build simulation engine.

Validated materials are supplied with pre-defined calibration information in the software, therefore, it is important to note that calibration is not a prerequisite for build simulation.

Still, dedicated calibration can improve the quality of simulation results (i.e., the results will be closer to the real result).

Calibration is explained in a separate guide. It is explained in a separate guide.
The exercise goes through the analysis considering that the material is already calibrated.

The Benefits of 3DXpert's Build Simulation

During calculation, the simulation results are available to the user, layer by layer, as soon as they build up.

The Build Simulation predicts where the material rises above the powder bed (it is called recoater interference) and so prevents the damage to the equipment and makes sure that the build will complete.

It helps to find areas with excessive stress and areas where plasticity levels are too high (this means that in these areas the material will fails and so cracks or tears will form).

It helps to validate your design of supports for the model and the position of the part on the tray

Simulation Server

The Build Simulation requires large computing power and often requires a separate, high-end computer. This computer is the Build Simulation Server.

For the specifications for the Build Simulation Server, refer to the BS Server installation guide.

Once the 3DXpert Build Simulation software is installed on such a server computer, every 3DXpert user (client) in the same network can use the power of this server computer to calculate the simulation.

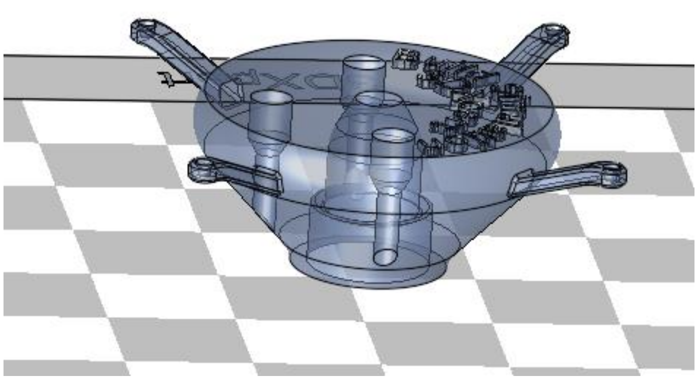
The system offloads your PC as the calculation is performed on the server PC, while the results are being sent back already during calculation.

About this exercise

The purpose of this exercise is to explain how to use build simulation and its various options. It does not explain how to setup a model for optimized printing results.

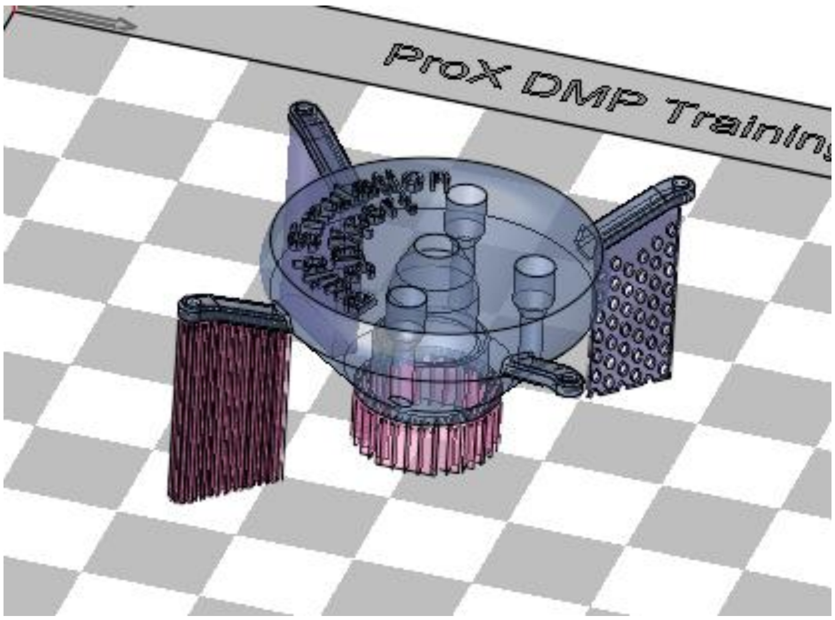
Running a Build Simulation

1. Load the project called 'Before Change.elt' (unpack the ctf file) and let's take a look at the part.
   This is the original, designed, model.

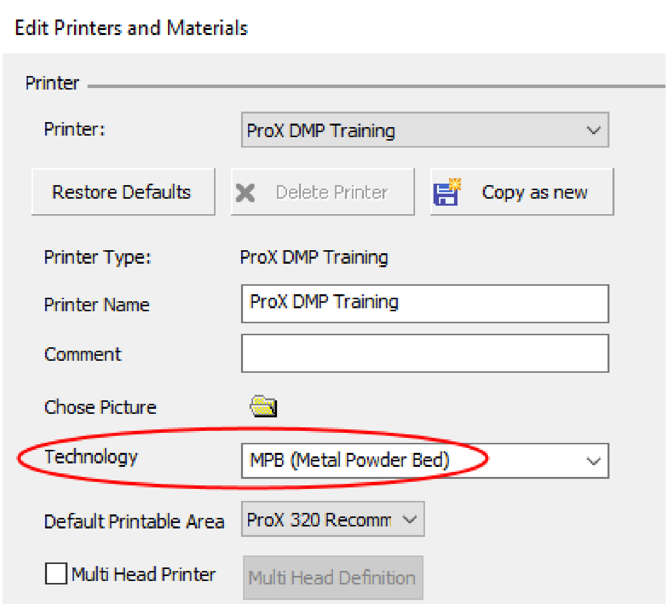


It has complex geometry, it has undercuts inside it.

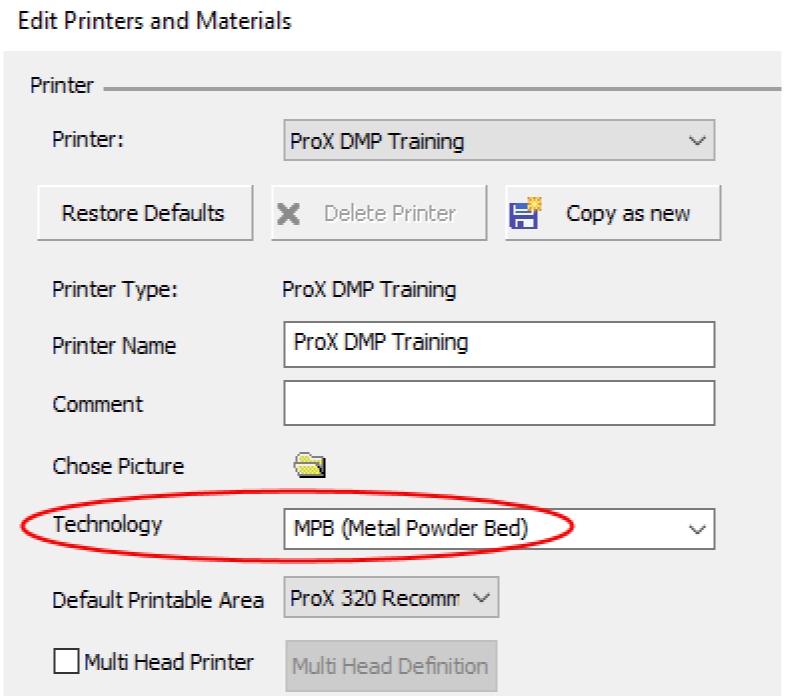
Note the four knobs. Clearly, these knobs require some support structures in order to withstand the printing process.
For the purpose of the exercise, we have created different types of supports: a **solid support**, a **wall support** with a pattern and a **solid wall** support. The fourth knob has been kept without a support structure, this will be used as a reference.



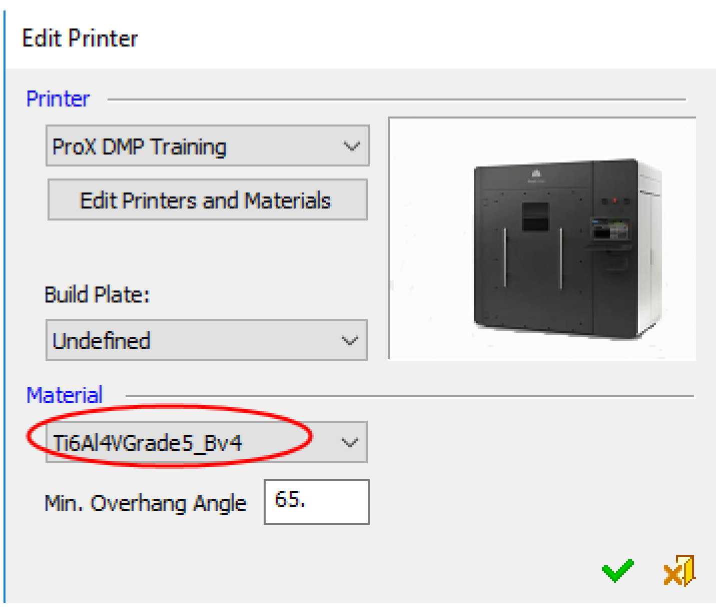
The **Build Simulation Analysis** and the **Residual Stress Analysis** functions are only available when the **MPB** technology is selected.

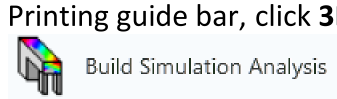


2. Edit Printers and Materials Parameters. Select the **MPB Technology**. Save the printer.

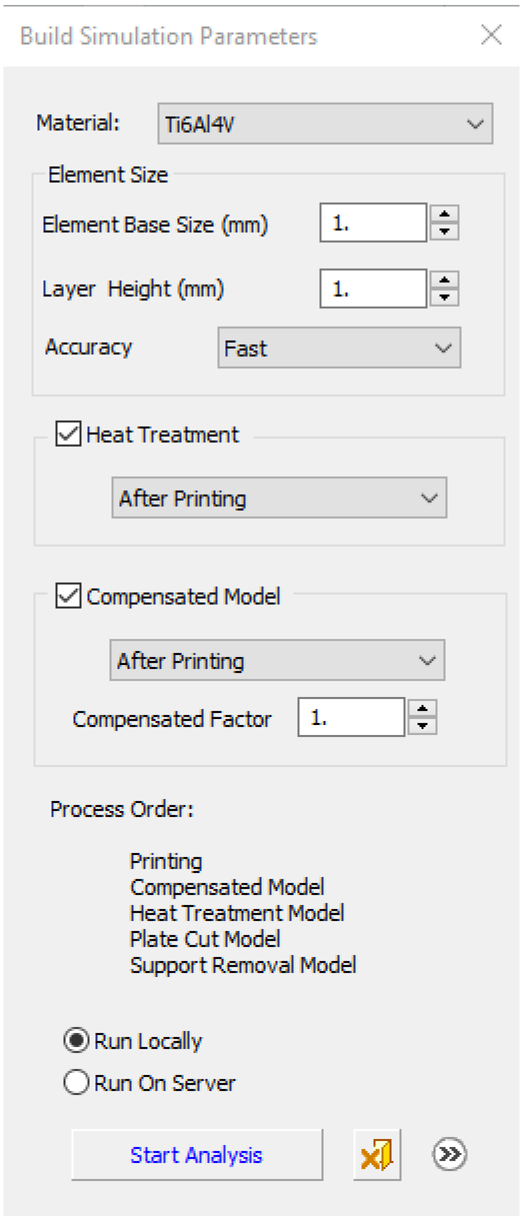


3. Edit Printer and select the **Ti6AL4VGrade5_Bv4** material. Save the printer

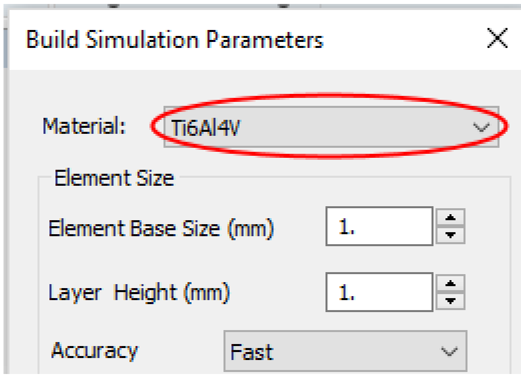


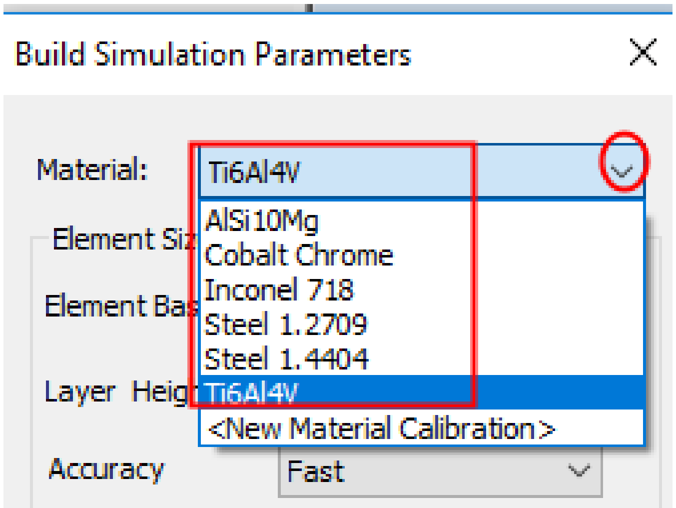
4. Now let's run the simulation. Through the 3DP Printing guide bar, click **3DP Analysis** and press **Build Simulation Analysis** 

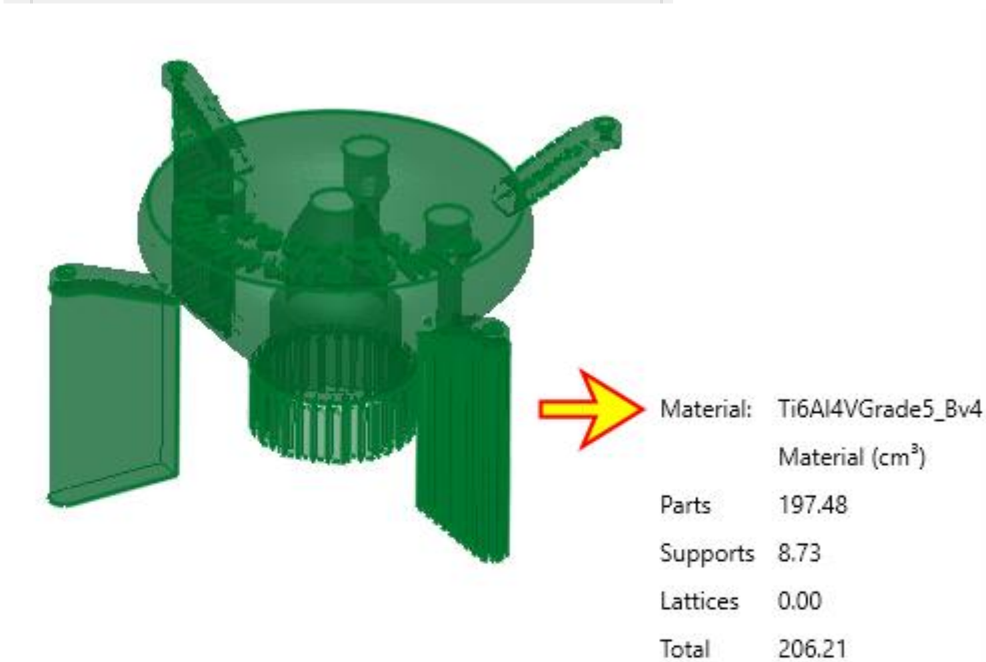
5. The **Build Simulation Parameters** dialog is displayed



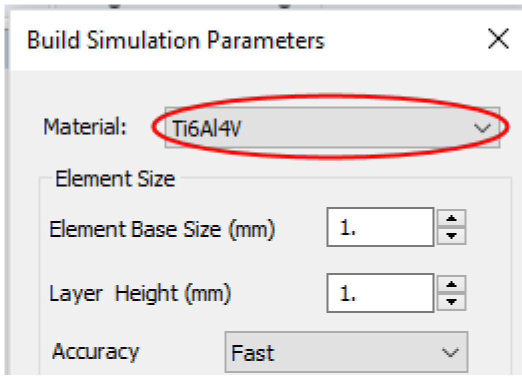
The listed predefined material type are used for the current simulation. The system associates the material name defined in the Edit Printer dialog, in our case: **Ti6AL4VGrade5_Bv4,** (which is also appearing at the bottom right of the graphics window), with a specific predefined material type. For example, if the material name is recognized as any type of titanium, the system will use the predefined generic titanium material (**Ti6AL4V**).

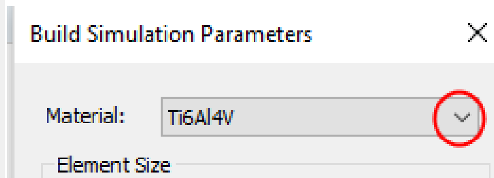




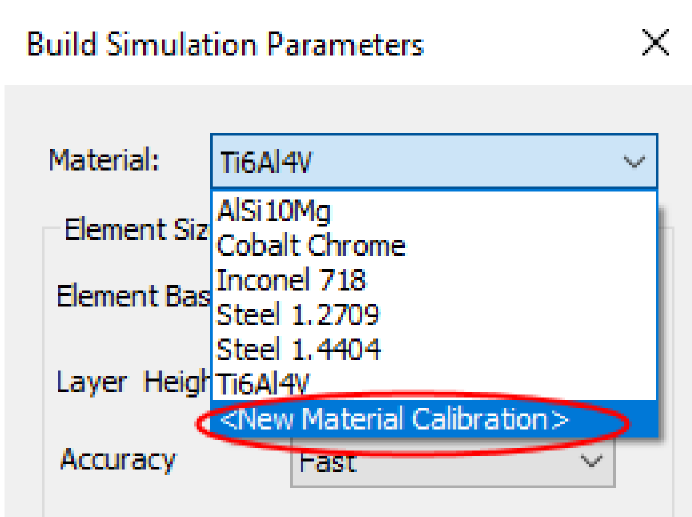


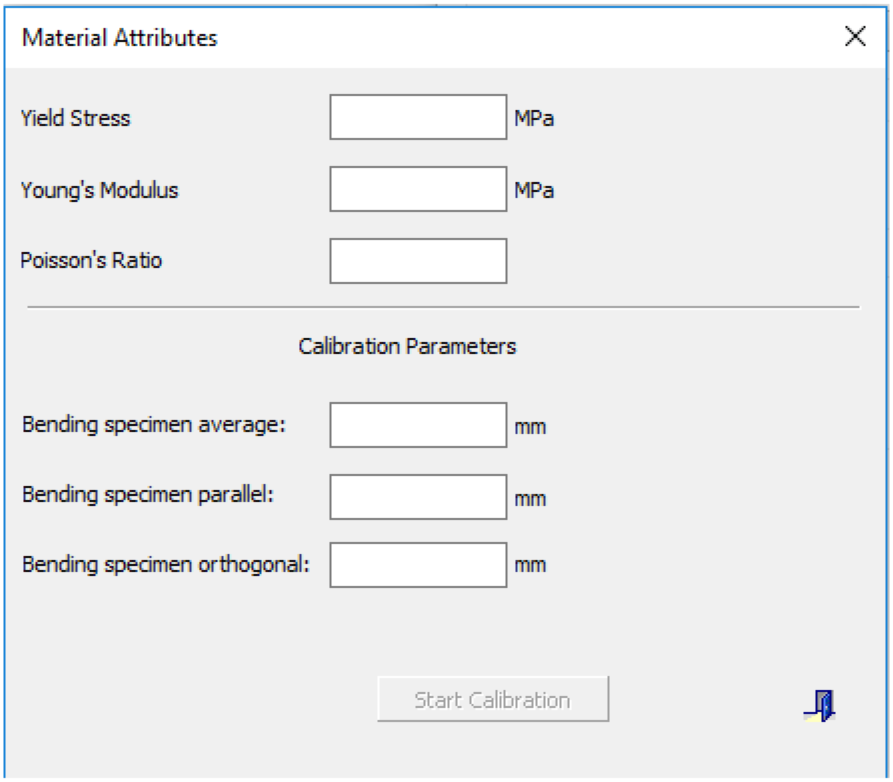
6. Based on the material used in the 3DP project, the system assign the matching material for simulation. If you are using a different material or wish to set another one, click the material. Accept the default generic **Ti6AL4V**.



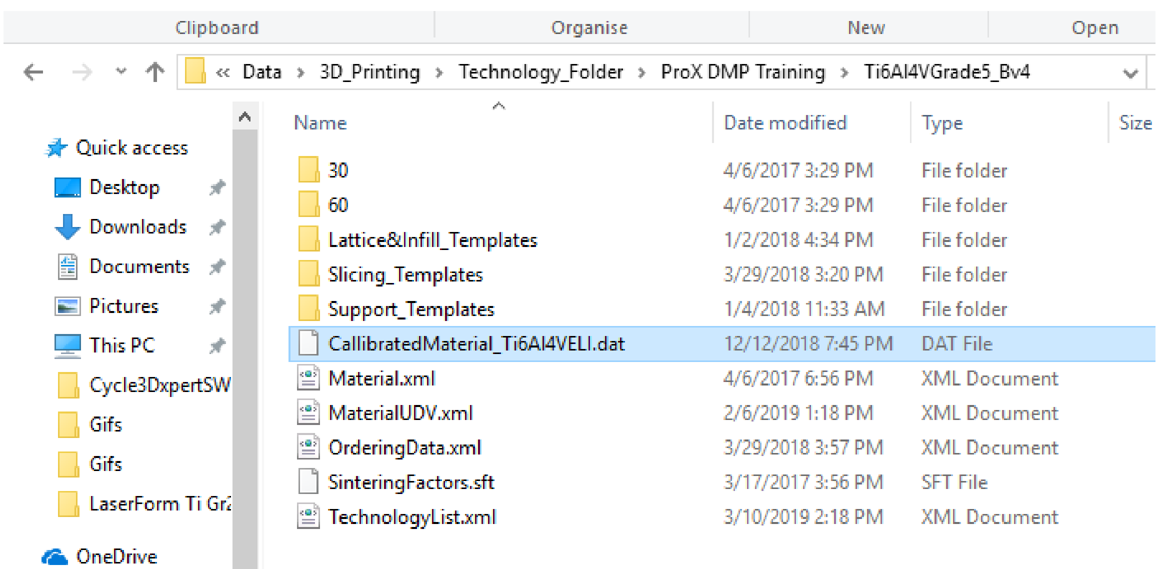


The last item on the dropdown list is **<New Material Calibration>**. Clicking this item opens the Material Attributes dialog, which enables you to run your own material calibration.





Once a calibration is done, a dat file is added to the material folder.

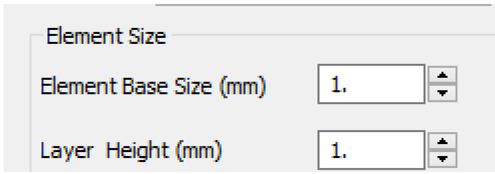


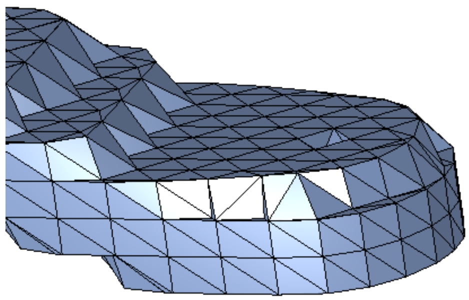
7. Go through the various simulation options. First, define the element size.
   Before starting the calculation, the system converts the model into small 3d element, in which:

   Element Base Size - the X (or Y) dimension of each element

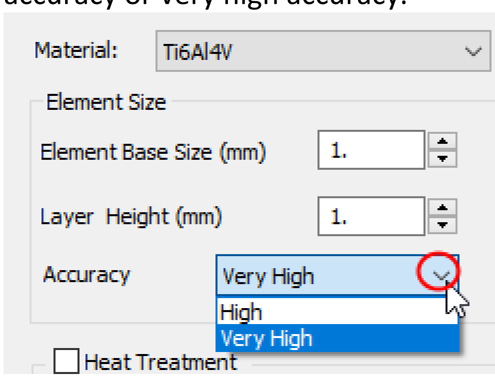
   Layer Thickness - the Element's height (Z) dimension of the cube.

Note: the height of the element has substantial effect on the calculation time.

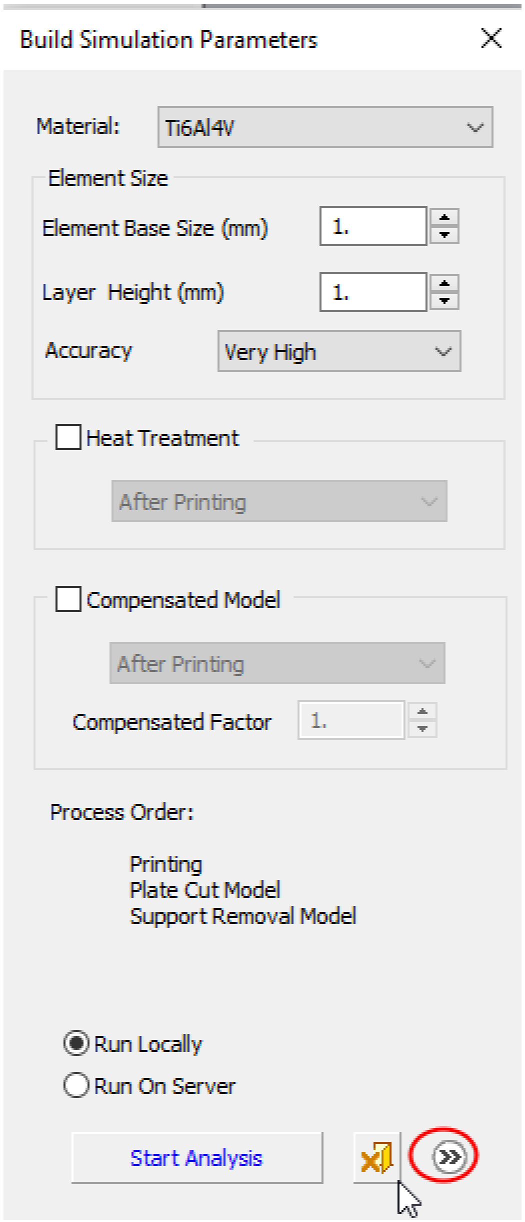


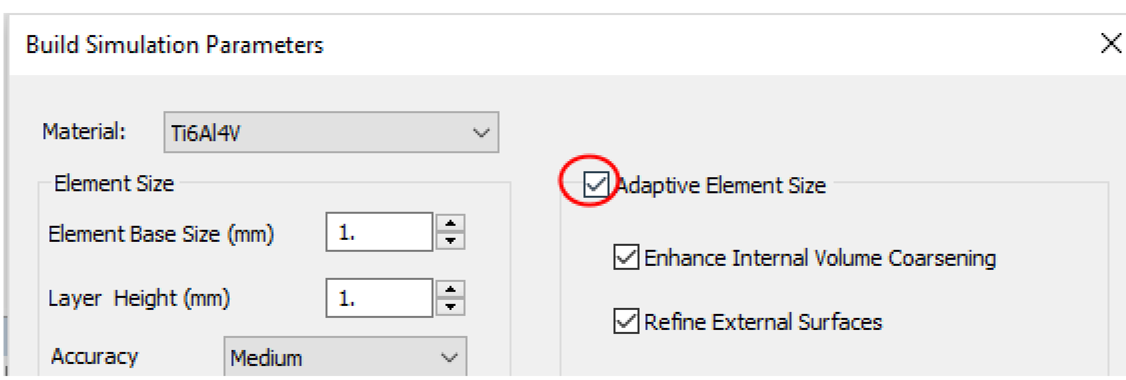


In refer to the element size as layer height; the calculation can be with high accuracy or very high accuracy.



8. Check the **Adaptive Element Size** in the expanded dialog and select the Fast accuracy option.





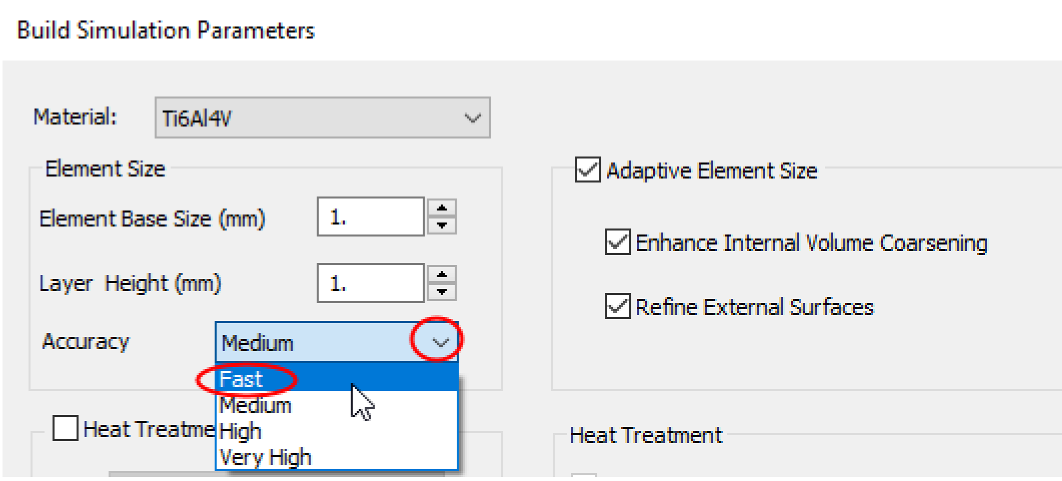
The Build Simulation Analysis enables you to automatically control and adapt the size of mesh elements between different areas of the part, to shorten the simulation calculation time. An example of adaptive mesh size would be finer mesh elements around external surfaces (where stress-induced structural fluctuations are more likely to occur) and coarser elements for the internal structure of the part (where any structural fluctuations would have less impact).

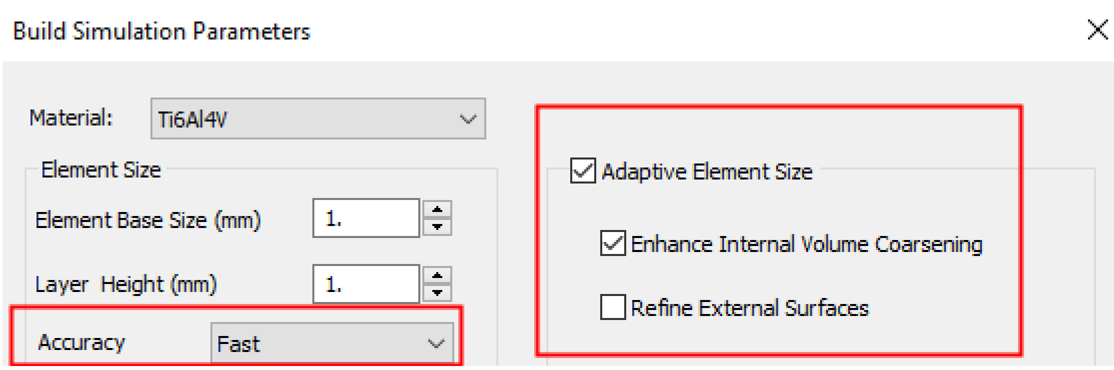
This option controls the adaptive meshing and is affected by the **Accuracy** level settings.

When this checkbox is OFF, the **Accuracy** level dropdown options are either **High** or **Very High**.

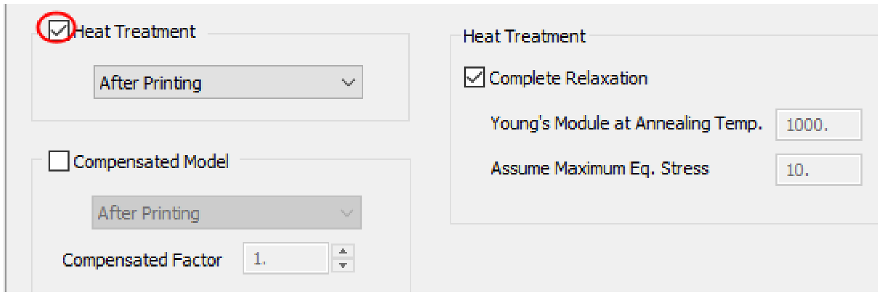
When this checkbox is ON this allows the adaptive mesh to combine more elements and thus to fasten the process.

9. Select the **Fast** accuracy. This turns **ON** the **Enhance Internal Volume Coarsening**. Turns **OFF** the **Refine External Surfaces**.



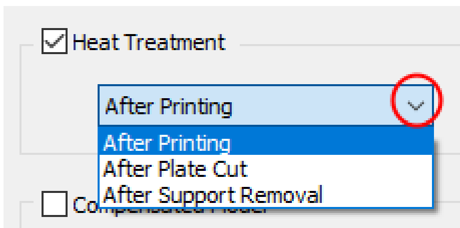


10. Next, define if to calculate also heat treatment.



Residual stresses induced by the additive manufacturing process tend to be very high, and therefore, some sort of stress relief treatment is often included.

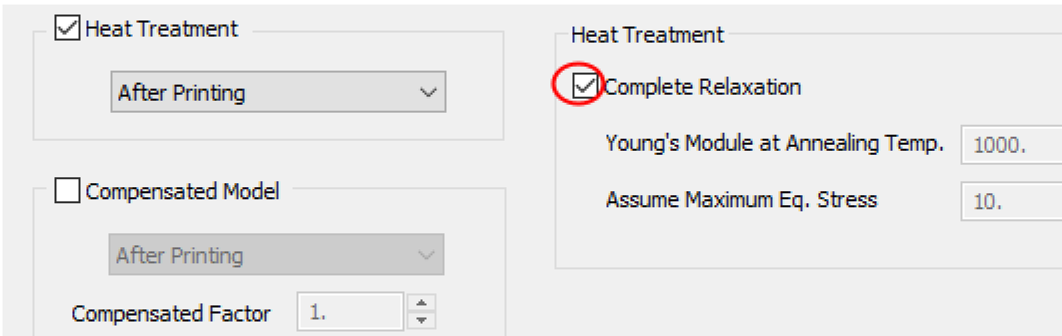
Heat Treatment can be calculated after any of the stages shown below.



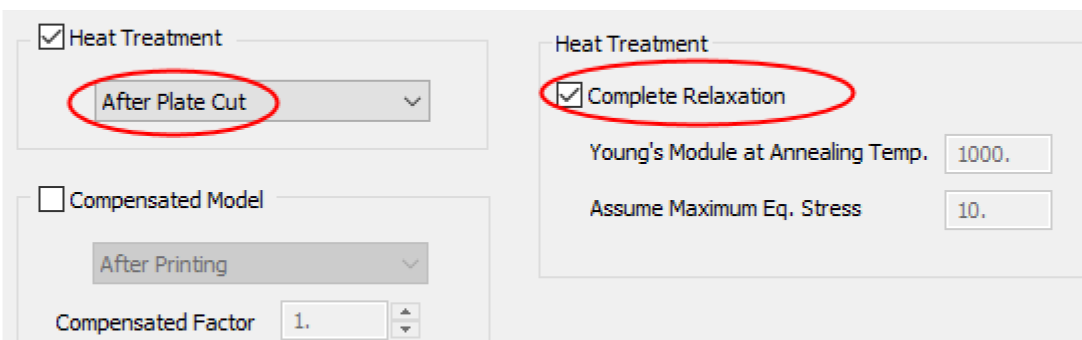
Once selected, this will be reflected in the the order on the results pane, so the order of the leafs in the tree depends in the order set by this parameter.

By default, heat treatment calculation consider that the model is fully relaxed. In other words, the stress relief treatment reduced the internal stresses within the part to zero, and so this no influence on the deformation of the specimen. This state is called Complete Relaxation.

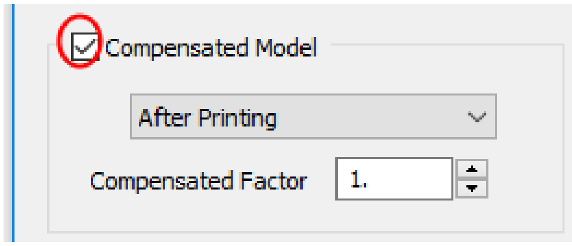
If this is not the case, and some stresses are expected to remain after heat treatment, then uncheck this option.



11. Set the **Heat Treatment** to be **After Plate Cut** and keep the settings as **Complete Relaxation.**



12. Next, set it to calculate the compensated model.

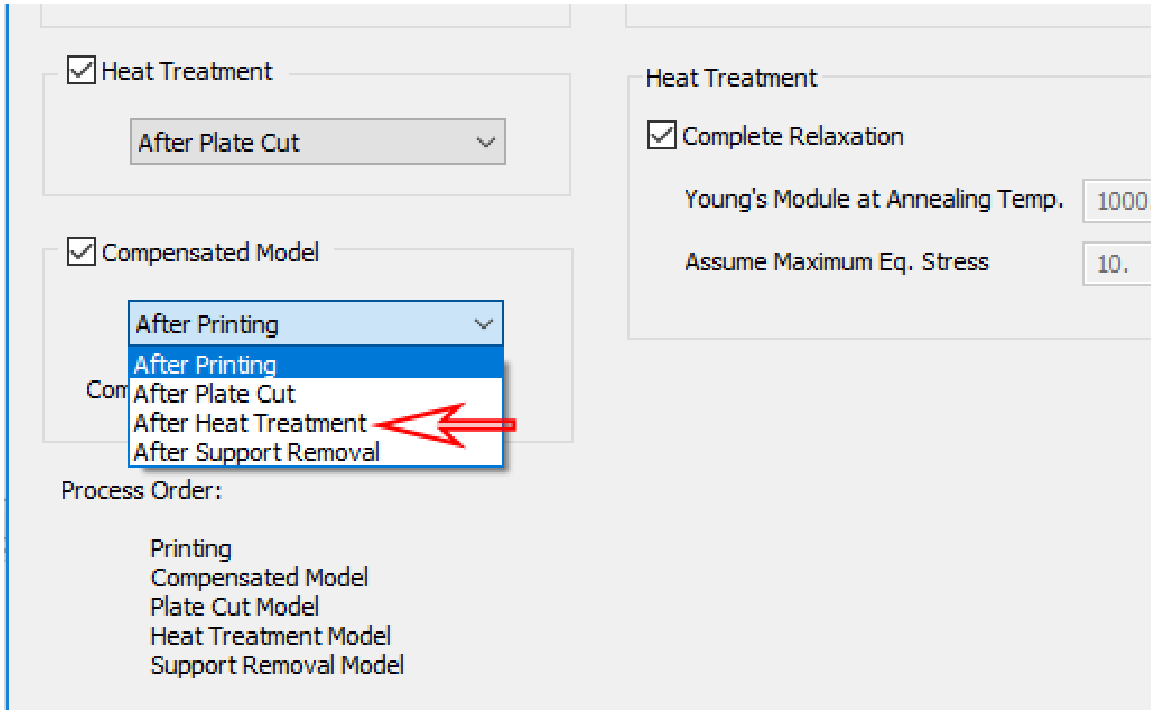


The compensated model represents how the model **should look like before printing**, so that it compensates for all the deviations.
Thus, once printed the result is a part that resembles the exact original model, as in the design.

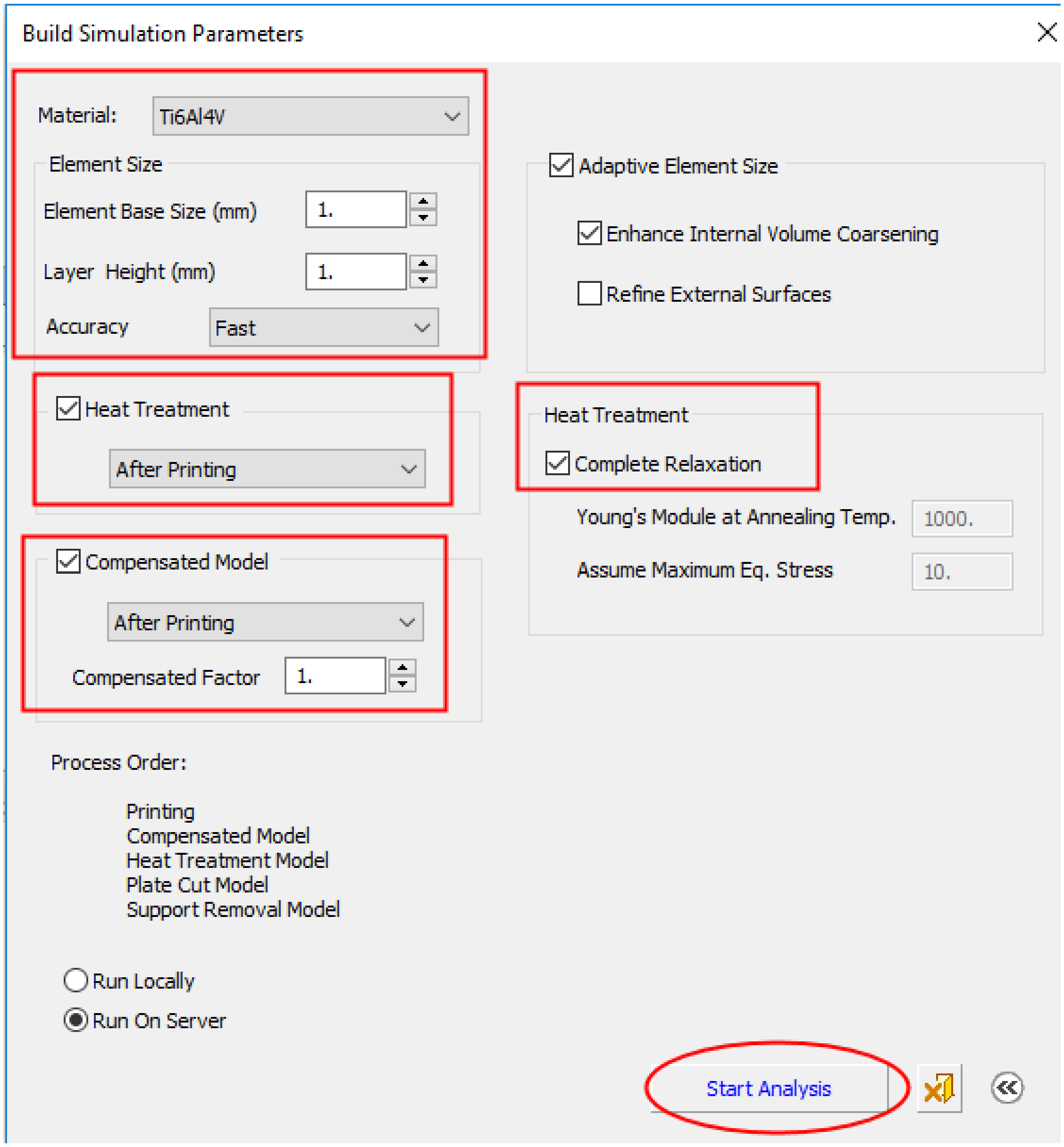
So now, set the stage in which you wish to take into account the compensated model (the deviation).
This stage can be calculated right after the print operation (at this stage, the part and supports are still connected to the plate), or the right after the part and supports are cut from the plate or even the after the supports' removal.
After each of these stages, the part will change due to the internal stresses.

In addition, you can calculate the compensated model after heat treatment (if heat treatment was calculated). Notice that in our case, the compensated model for After Heat Treatment appears directly after the compensated model After Plate Cut appears.

This is because we previously set the Heat Treatment to come right After Plate Cut.


Heat Treatment
After Plate Cut
Compensated Model
After Printing
Com
After Printing
After Plate Cut
After Heat Treatment
After Support Removal
Process Order:
Printing
Compensated Model
Plate Cut Model
Heat Treatment Model
Support Removal Model
Heat Treatment
Complete Relaxation
Young's Module at Annealing Temp.
1000.
Assume Maximum Eq. Stress
10.

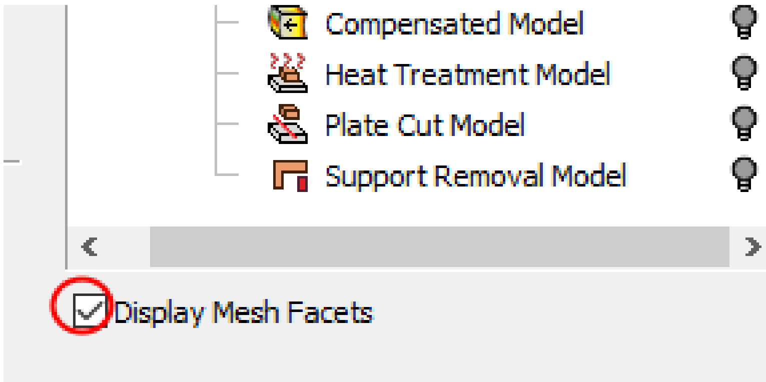
13. Finally set the parameters set as below.

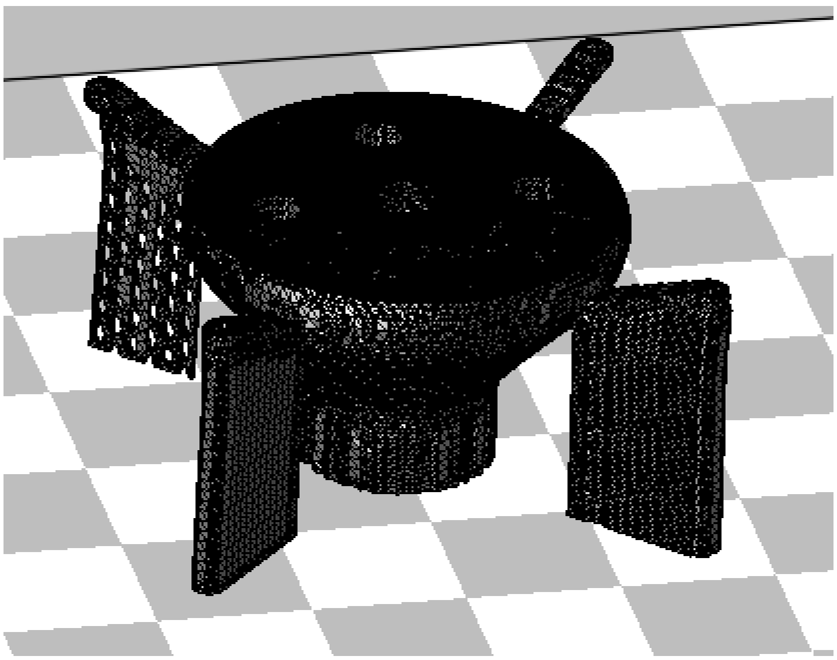


14. Press the **Start Analysis** button and the calculation takes place.

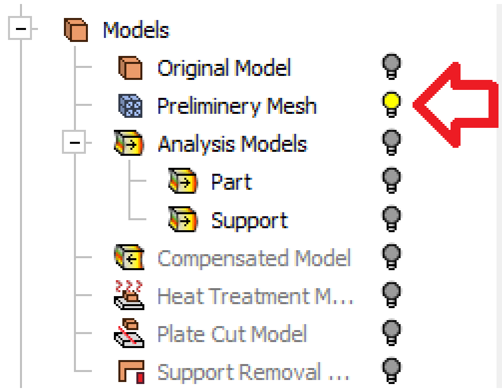
Note: this may take time, depending on the hardware in use.

The first stage is the meshing stage (the system is turning the model into a model, based on the element size. Check the Display Mesh Facets option.

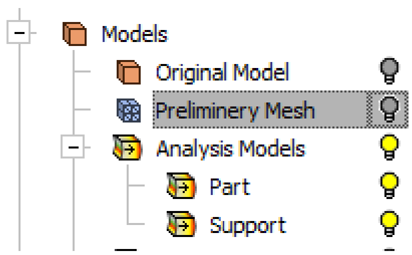

Compensated Model
Heat Treatment Model
Plate Cut Model
Support Removal Model
Display Mesh Facets



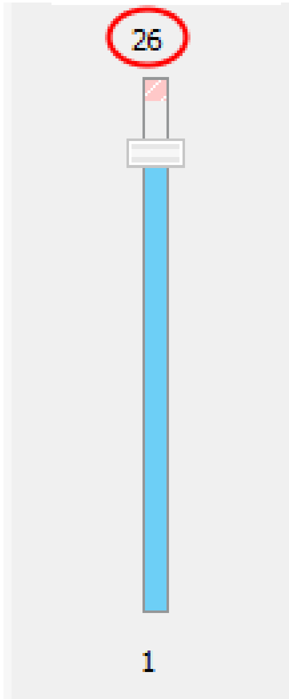
You can see this model only by switching off the original model and displaying only the Preliminary mesh through the Build Simulation Tree



15. Once meshing stage is over, the analysis starts to run.
    The simulation result are dynamically updated during calculation, layer by layer.
    Switch off both Original Model and Preliminary Mesh and show the Analysis Models:

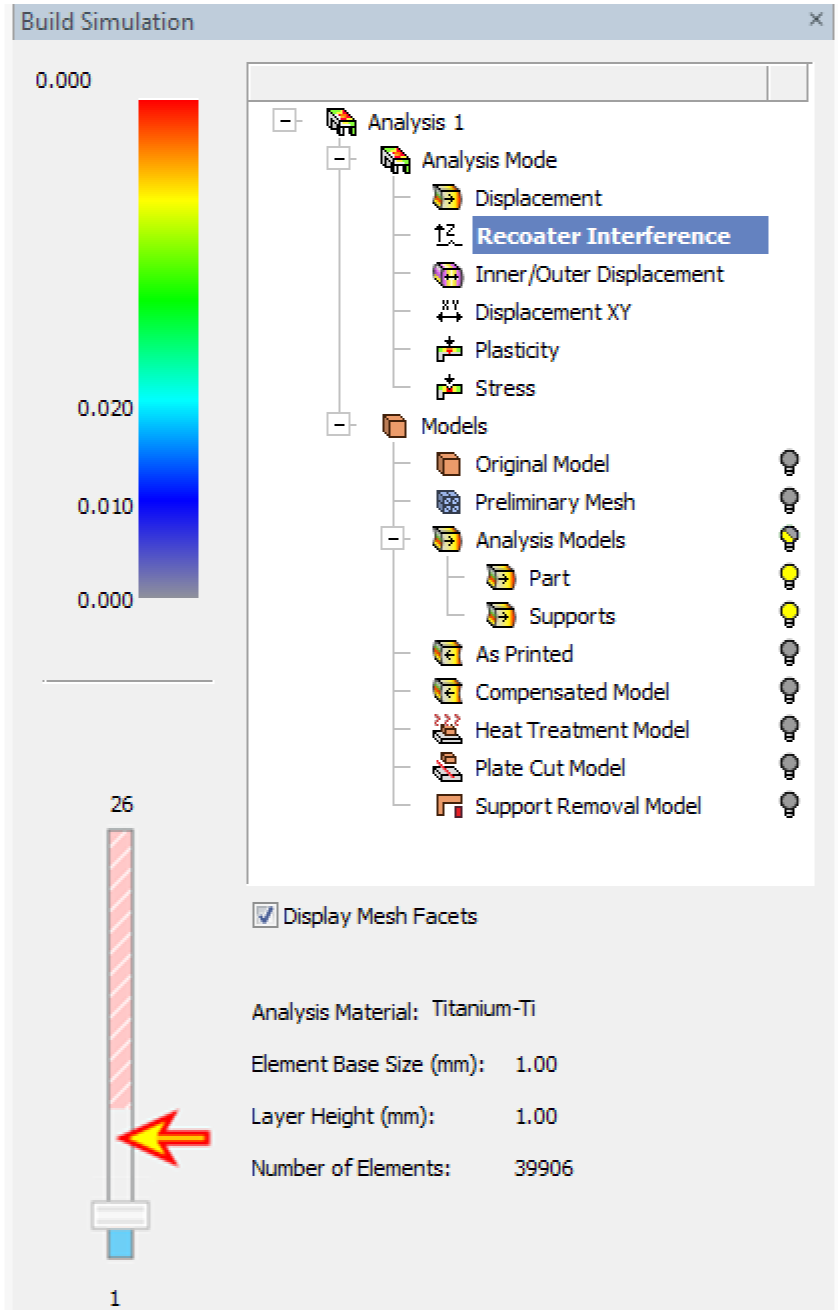


Note the vertical bar and see the number 26, which is the number of layers.



Number of Layers

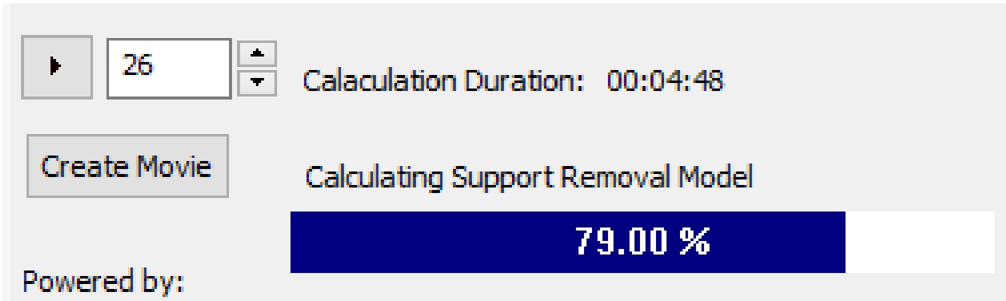
> **!** NOTE: While we explain here the calculation stages, there is no need to wait for calculation to end. You can already examine the result. Continue with this exercise - step 9 onwards - as the calculation is running.



The red area of the bar indicates the layers that still need to be calculated.

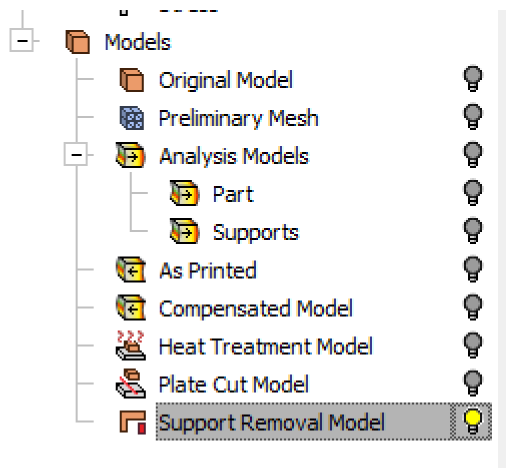
At the bottom, below the calculation bar, notice the number of layers which were calculated so far.

In the next stage, the system calculates the results for each stage



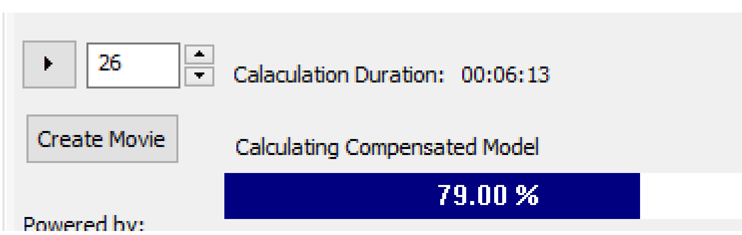
Notice the display mode ON/OFF for each of the models

(Heat Treatment, Plate Cut & Support Removal models).



You can display the models during calculation.

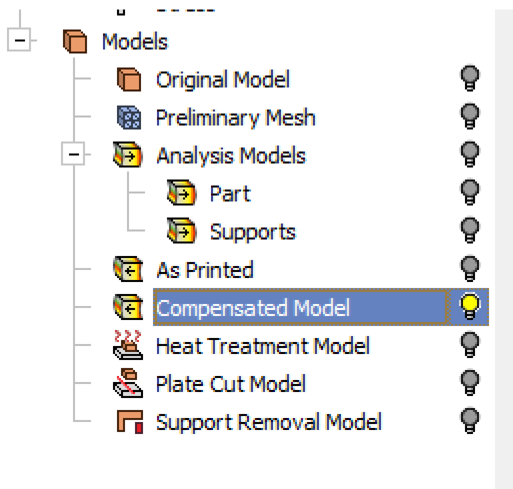
Finally, the compensated model is calculated



Once finished, the compensated model can

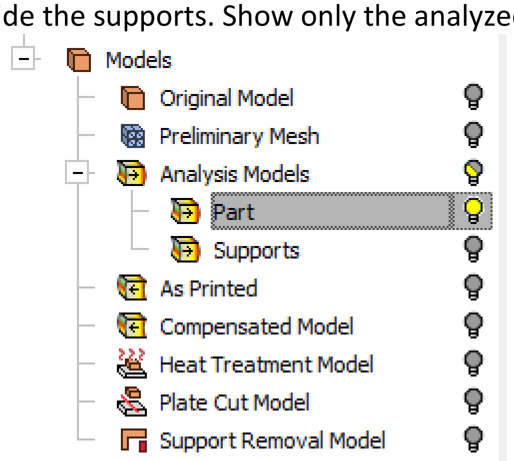
also be shown or hidden via its bulb

appearing on the tree.

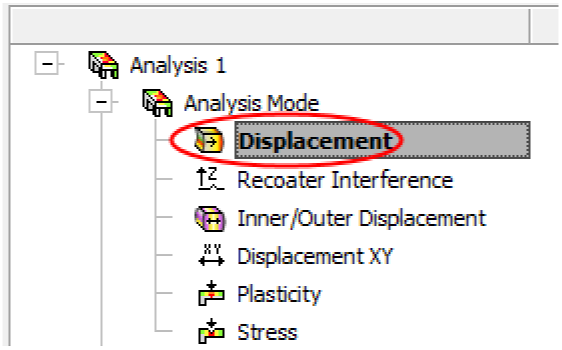


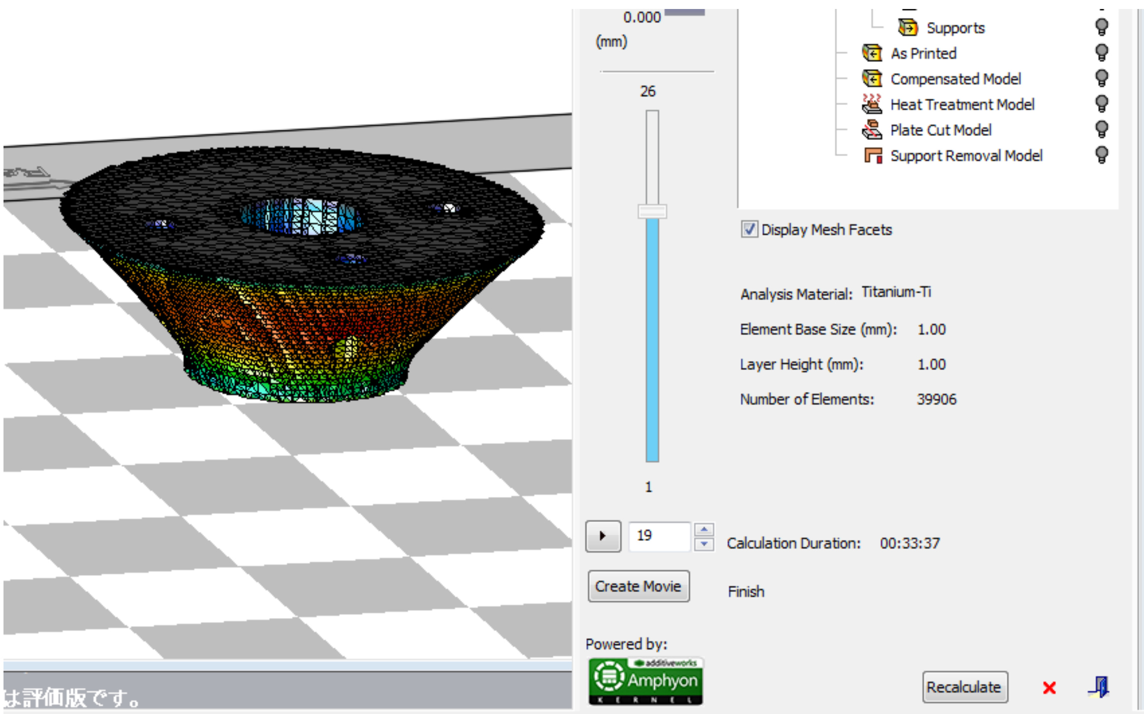
At this point calculation is over. Let’s examine the results.

16. Hide the supports. Show only the analyzed part:



17. Click the Displacement leaf on the Build Simulation Tree, pick the simulation bar and drag it upwards to see the displacement along the layers that were calculated.



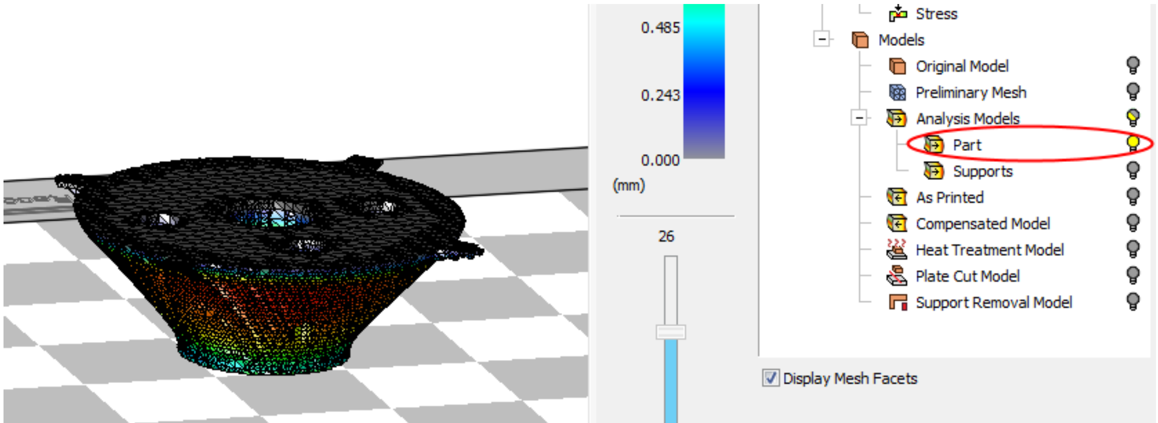


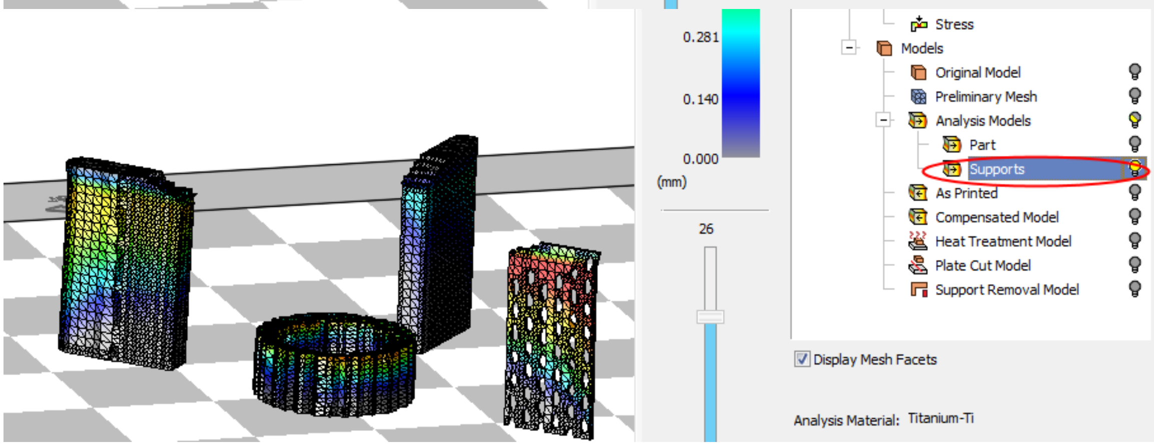
Displacement

The **Displacement** map indicates the deviation of the original model – the overall displacement in X,Y,Z directions of each point on the model (in mm).

Notice the peak value. The color map manifests the displayed models.

Now hide the part and show the supports and notice that the peak level changes accordingly.



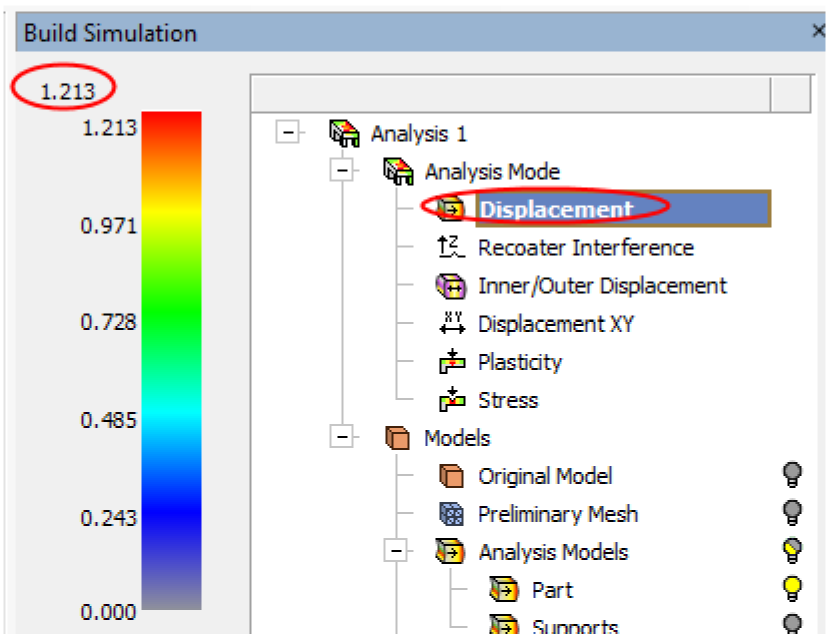


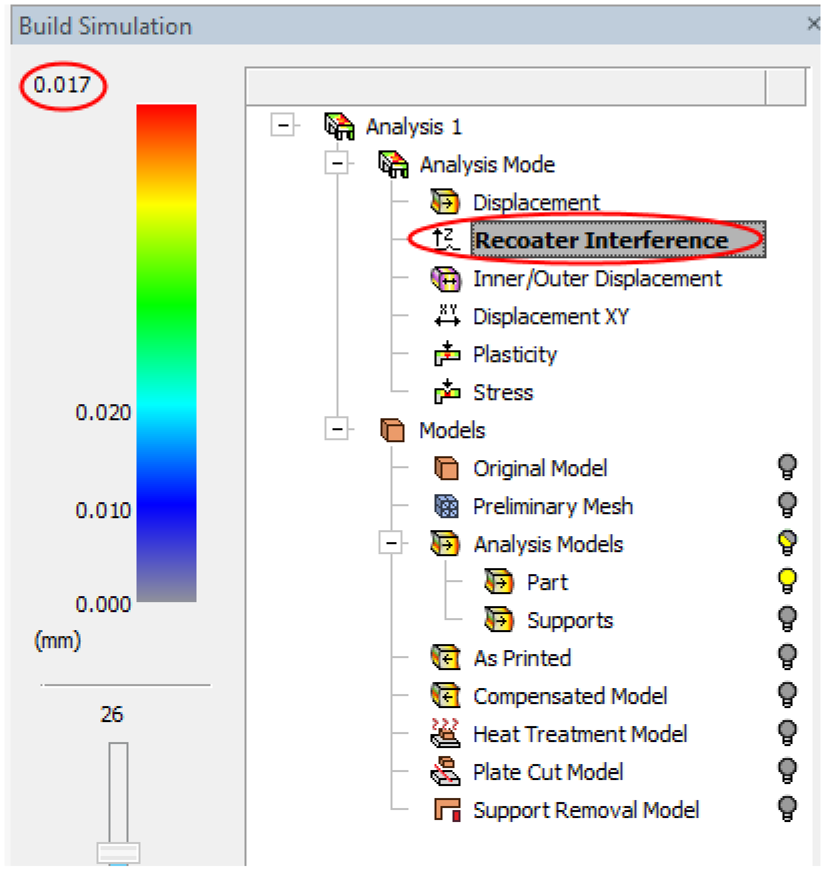
Displacement – Part Only　　　　Displacement – Support Only

Notice that the color maps are similar, but the values showing up along the colors are different, as the range consider the displacement of the displayed models.

The same is true between the different types of analysis. As in any FEA analysis, the color scale is dynamic, a red in one scale and a red color another scale do not represent the same value.

Note here that the peak values for each of the analysis is different, although the colors maps are similar in each analysis.





18. Let's review the other analysis types:

    The **Recoater Interference** analysis shows how much material, as it cools down, exists above the powder bed. Above a certain value, this may become a serious issue and damage the recoater.

    Therefore, this analysis checks for the displacement in the Z direction. In general, starting from 0.02mm is where this may need attention (color is blue).

    The **Plasticity** analysis predicts plastic strain*, or the tendency to develop tears and cracks in the part. It should therefore be as close to zero as possible and definitely below 1 percent (0.01).

    ***More on Plastic strain:**

    Strain is not measured with units. This is a ratio indicating how much the material gets longer. A zero strain value means that the size of the part does not change under the conditions. It remains as before.

    The plastic strain is the portion of the strain which is above the elastic elongation of the material (or above the yield stress).

    The **Stress** map indicates the stress levels on the part and support (in Mpa).

19. If you have waited for the simulation to finish, the following models are now calculated and available for display:

    The **Plate cut model** shows the deformation that the part will have, after it is cut off from the plate.

    The **Support Removal Model** shows the deformation that the part will have, after the supports are removed.

    The **Heat Treatment Model** shows the deformation that the part will have, after the heat treatment.

    The Compensated **model** is the model that we will need to print and compensate for all deviations in order to get the required (as designed) result at the end.

    Remember that you can hide and show the different models from the tree in order to see the differences between them.

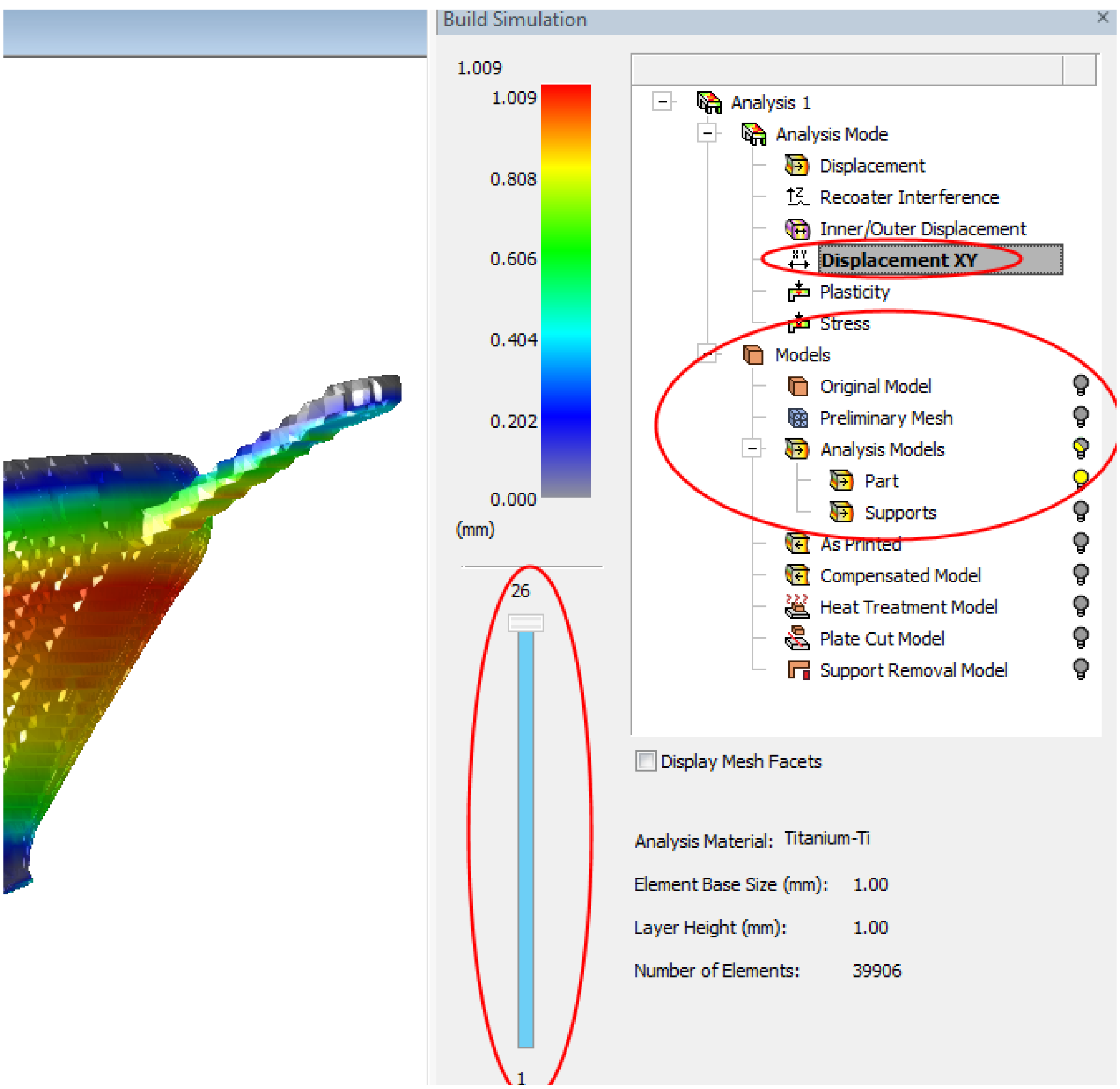
Analyzing the Results

Once the calculation is over, review the results.

20. Click on the tree to show **Displacement XY** - Notice the largest displacement (in red and yellow) where we did not place any supports.

    Naturally, being thinner than the part, the wall support is less susceptible to stresses.

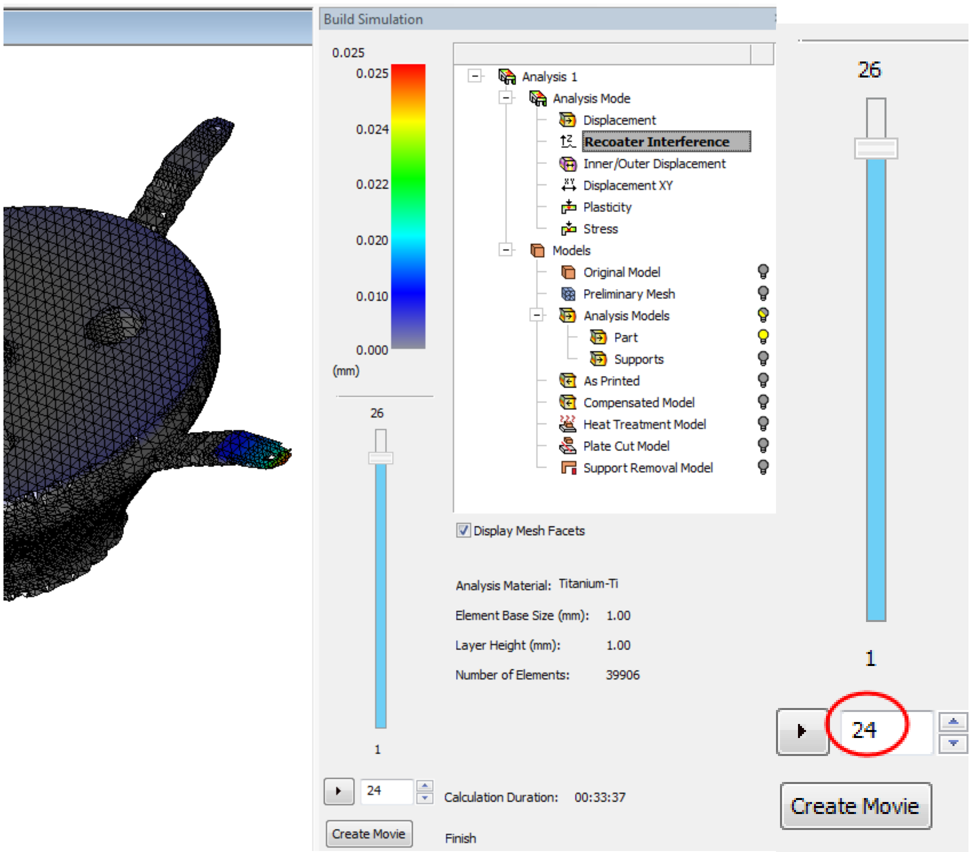
    Note to see the whole part (as below), you need to show the part and drag the slide up to the top



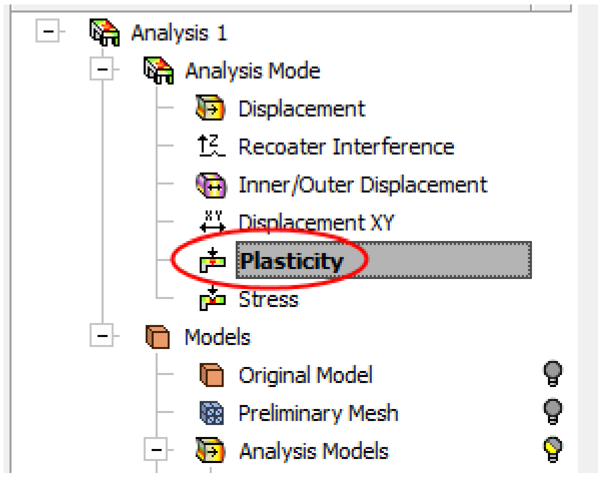
21. Click **Recoater Interference.**

This is an important check, as such a problem may even damage the equipment (hardware).

22. As you move between the layers, see that the outer areas are colored blue, also as the infill is building up. There may be problems at the top where there are no supports



23. Next, click **Plasticity.**

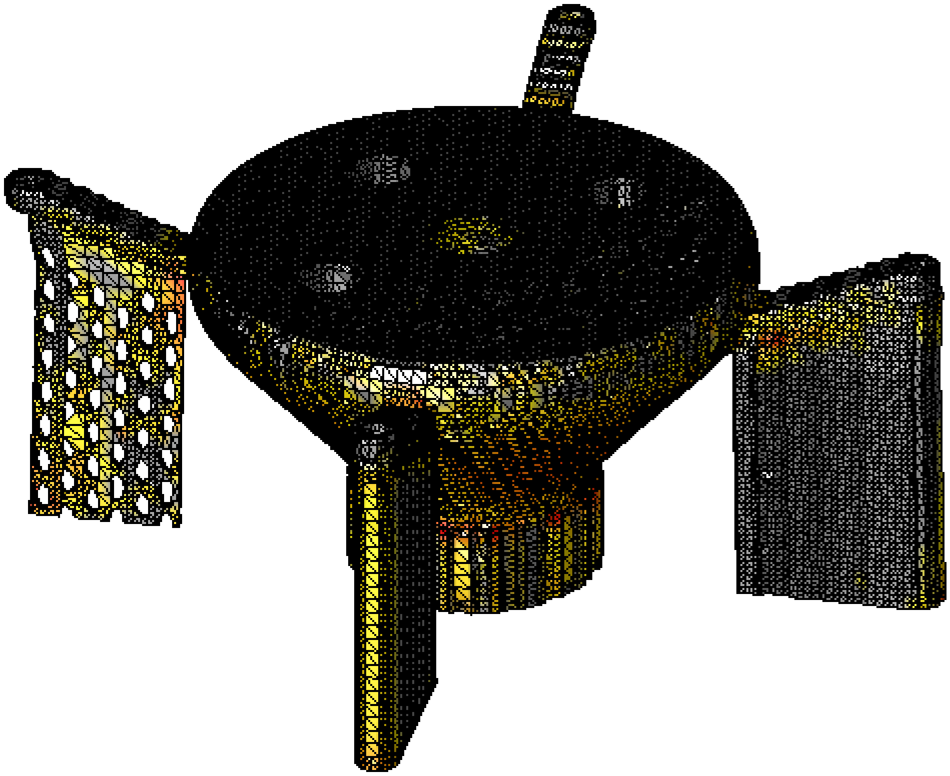


Now can verify the strain accumulating on the part and the supports as you go up. Note the amount of strain on the solid support

(this support was added here as an exercise)

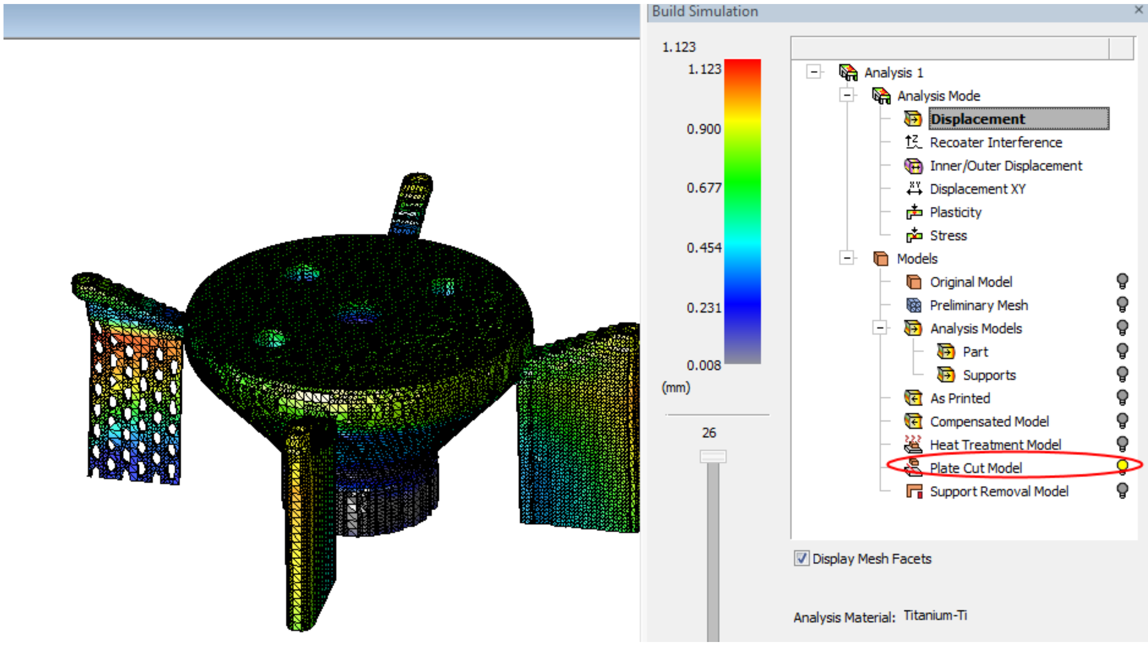
Cleary, this amount of strain will require attention.

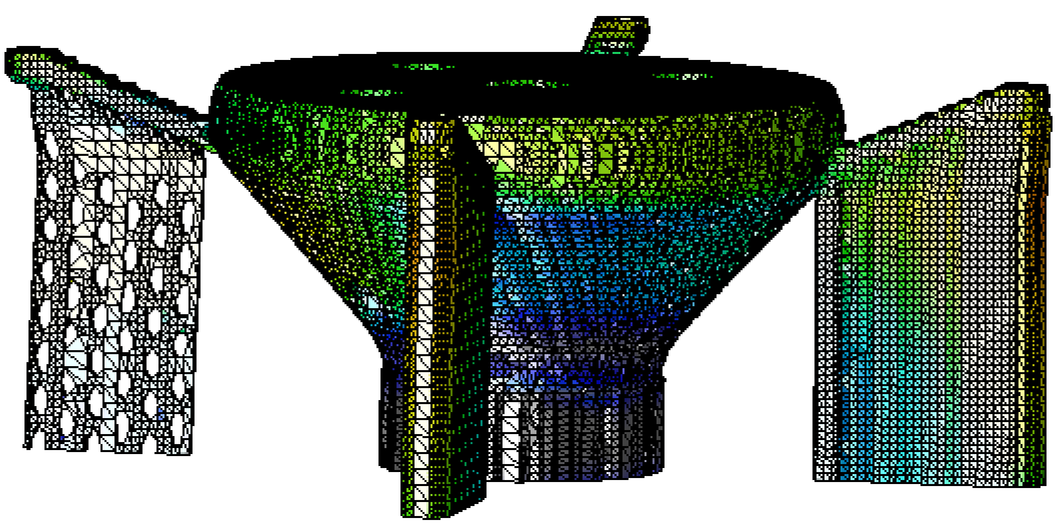
Note: At any time, you can display the mesh facets, this may help to visually check the results.



A solution might be to add wall supports that will help transfer the residual stresses and the heat.

24. Check the models.
25. On the Tree, switch to **Displacement** and click **Plate Cut Model** to view how the model may look like, after we disengage it from the plate.

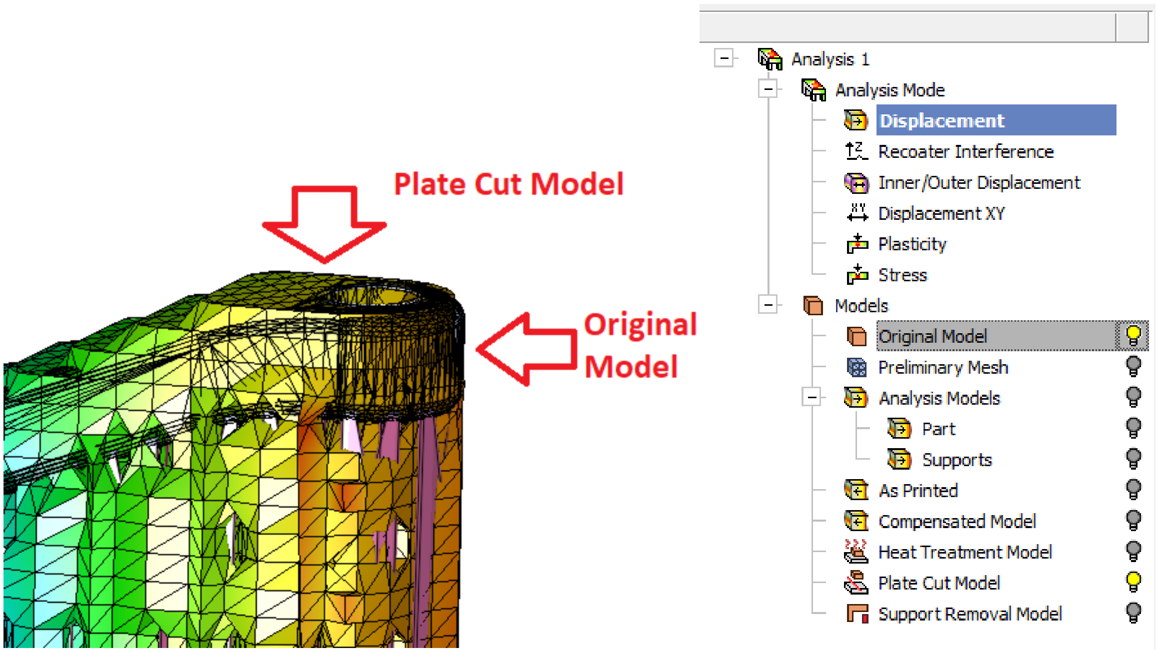




Notice how the supports structures are warped upwards.

You can visually compare between the original part and the Plate Cut Model (or the other models).

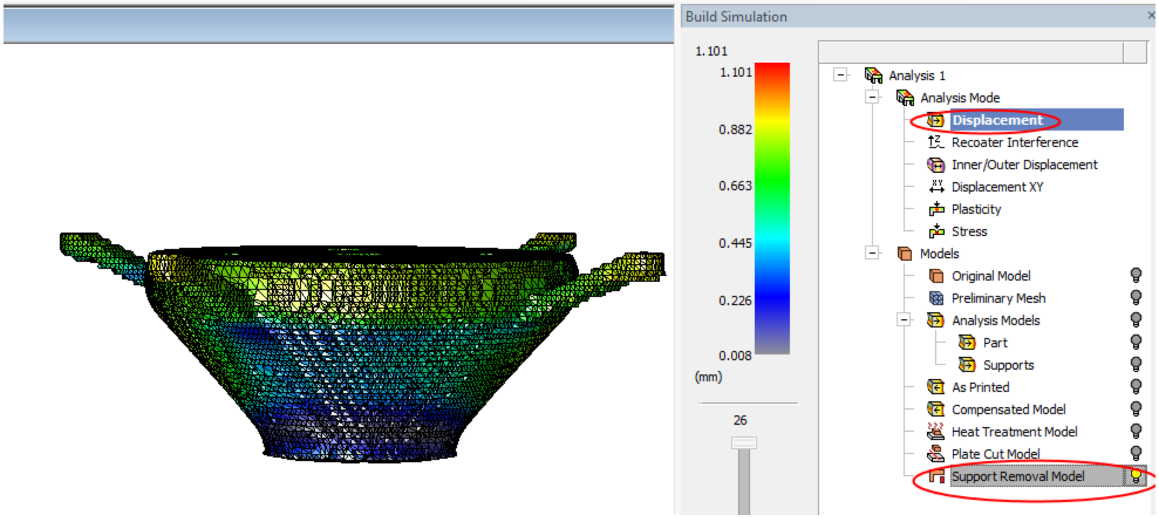
26. Click the bulb adjacent to Original Model.

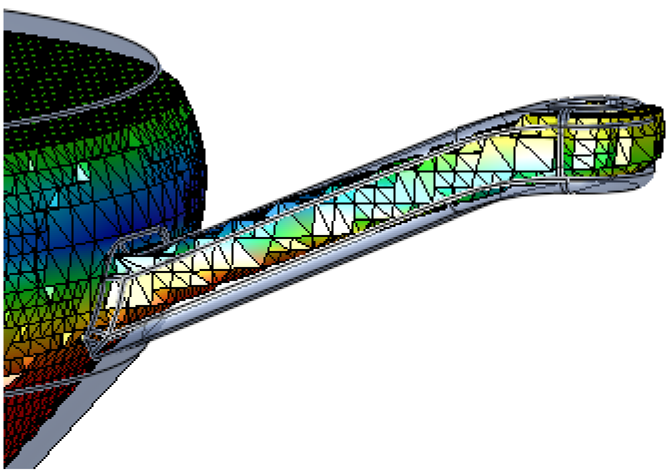


Next, review what happens after the supports are removed.

27. Keep the analysis is at **Displacement** and click **Support Removal Model**.

You can see that the knobs are shifted dramatically from their original position.

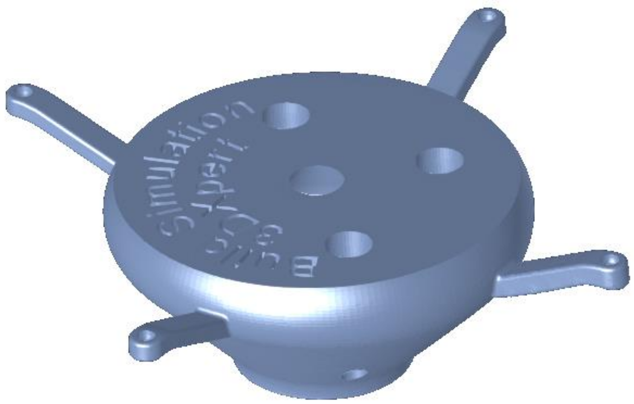


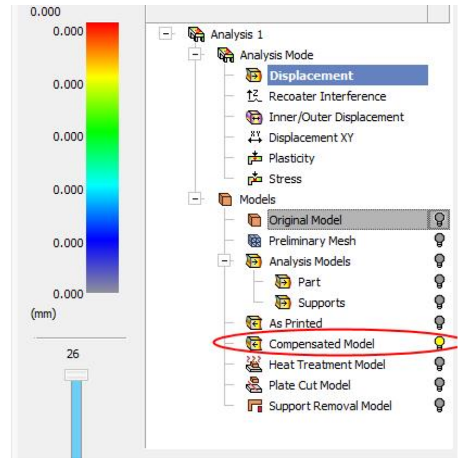


The Original Part & Support Removal Model

As noted before, in each analysis the colors refer to different values. Check the color scale for the actual value. A red zone in one scale is not the same as a red zone in another scale.

28. Remember that we have calculated also the Compensated Model.
Click to show the **Compensated Model.**



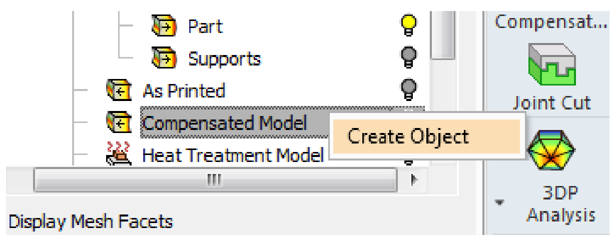


Reminder:

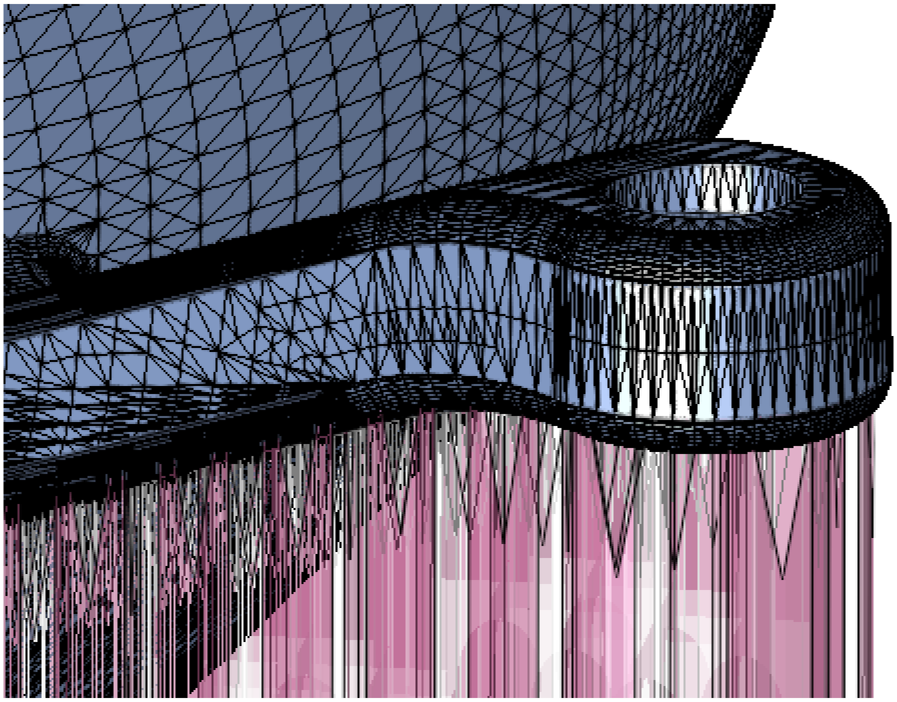
This model is a deformed model that you would need to print, in order to get the designed part.

What you see is a preview of the model. In order to work on this model, an actual object should be created.

29. Right mouse click the Compensated Model node on the tree and press Create Object

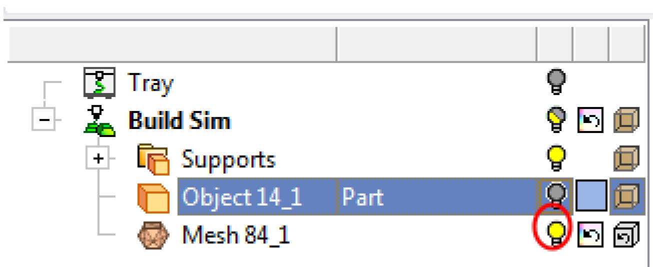


This operation creates a new mesh object, which is added over the existing model (both can be displayed)



This new object is not added on the Analysis tree.

30. To hide or show this object, click the bulb next to the new mesh object on the Objects Tree.



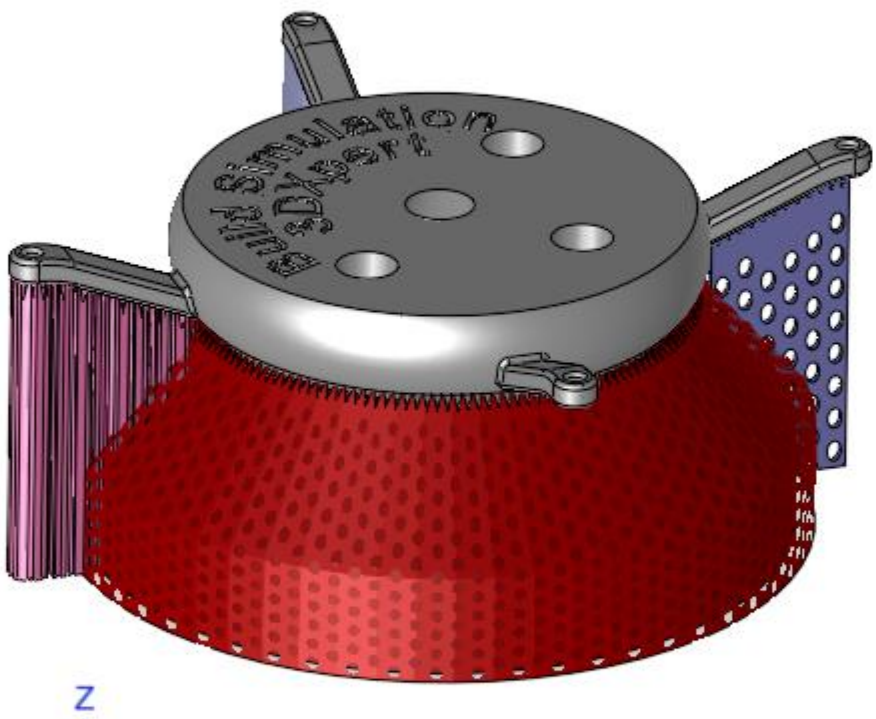
When you print the compensated model, the support strategy should remain exactly the same, as in the original design.

Simulating a Modified Model

So far, you have simulated the designed model and found some issues.

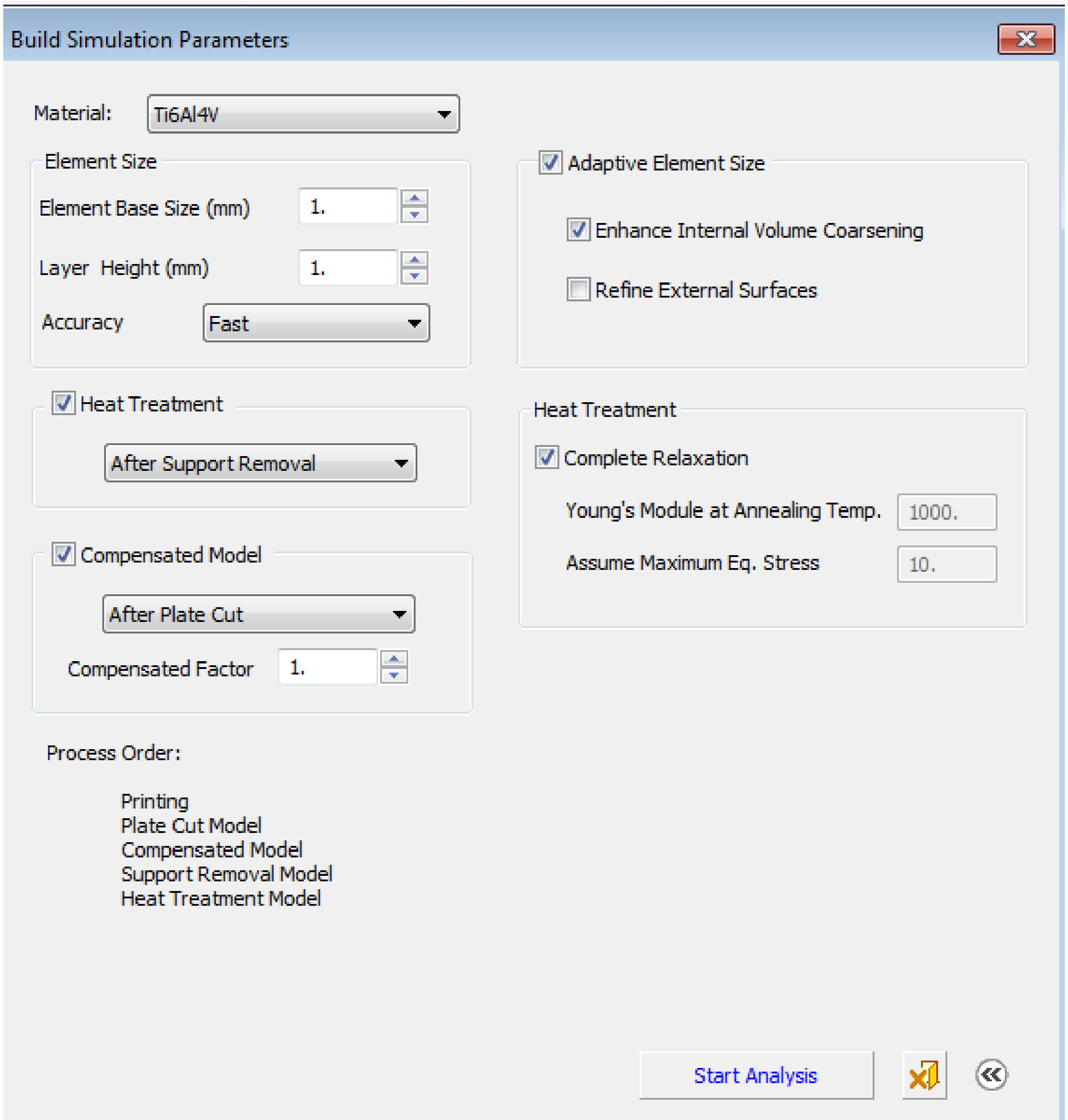
The next step was to update the design for printing and simulate again, in order to see if the results are better.

31. Unpack the After Change.ctf . Load the updated part 'After Change.elt'. This is a modified design.

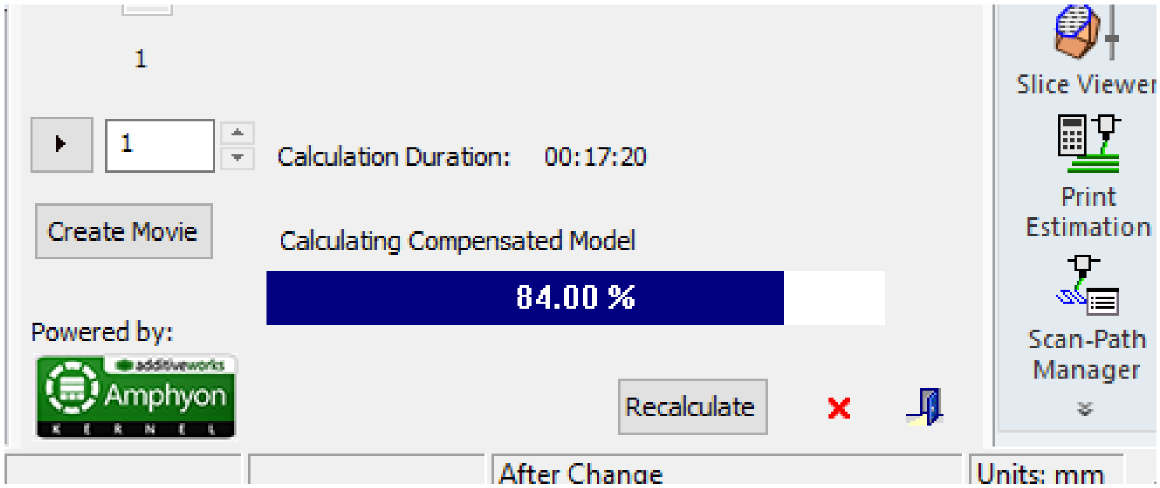


To improve printing results, the user added wall supports running around the model, using offset type pattern with a radial tilt to better transfer the heat and the accumulating stresses.

32. Enter **Build Simulation Analysis** and perform the simulation using the same parameters. (Recalculate)

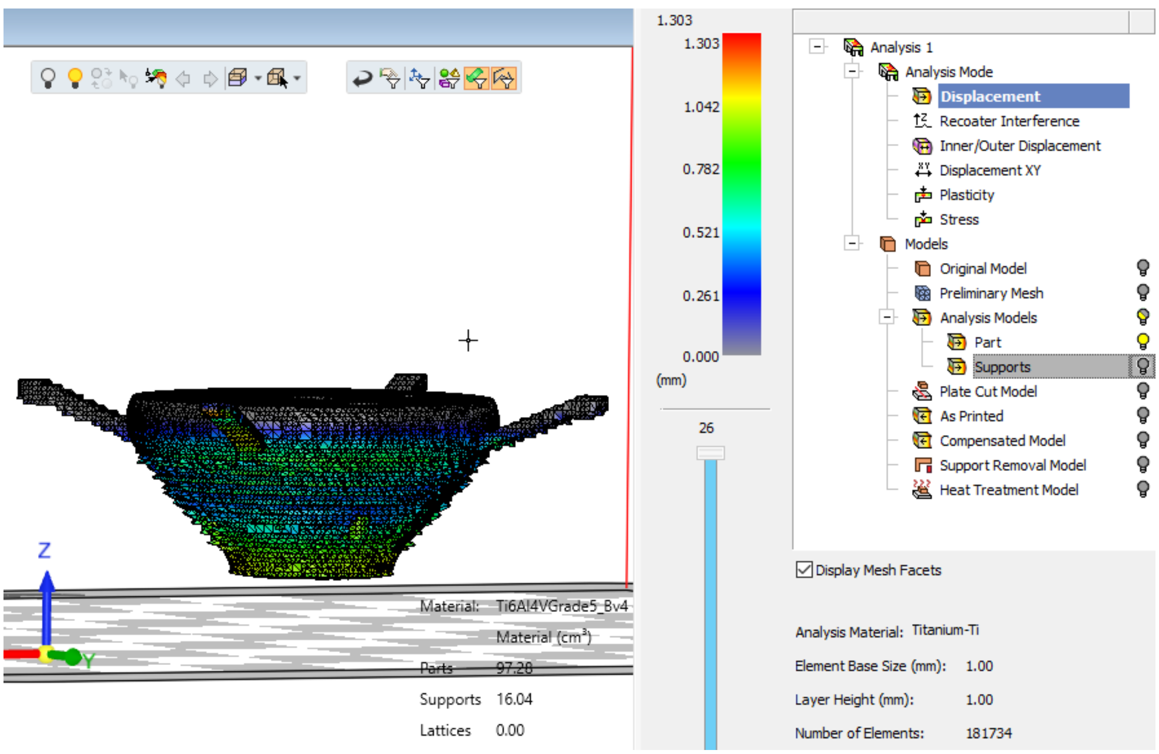


This may take longer as this model has a higher number of elements.



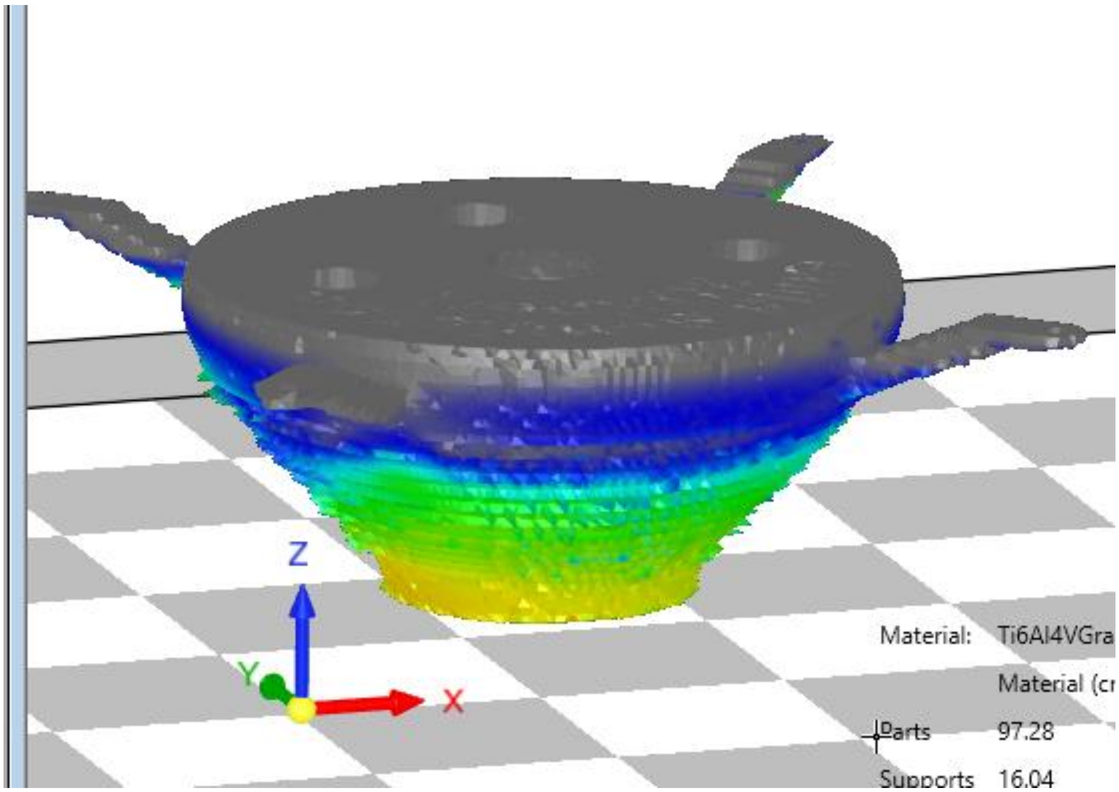
Once the Simulation is over, review the results.

As expected, there is a marked improvement in the results as you go up the layers.

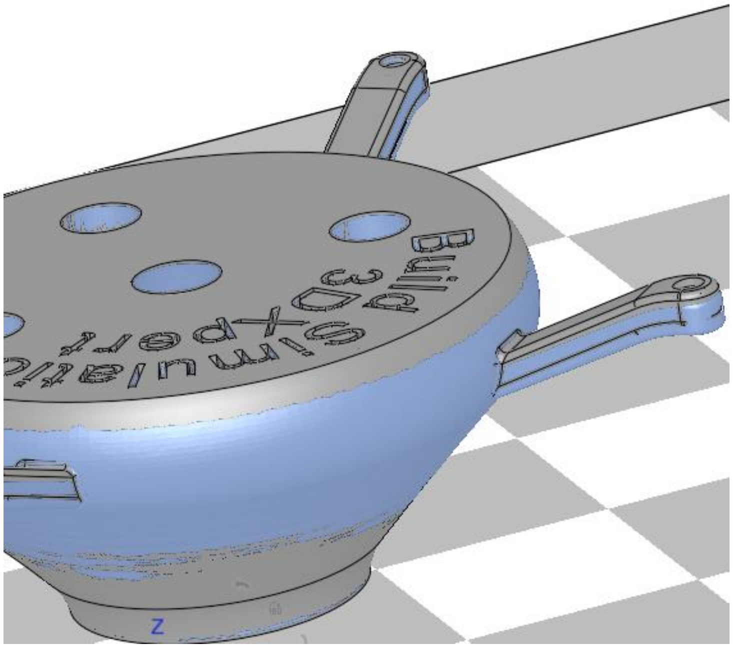


C

The amount of strain, for example, is much lower than before and the values are more or less constant as you go up.



The original model (in grey) and the compensated model (in light blue):



End of Exercise