ANSYS AdditivePrintTransfer Extension for Simulation of Cutoff

Use these instructions to use Workbench Additive for simulating after-cutoff results from a simulation performed in Additive Print.

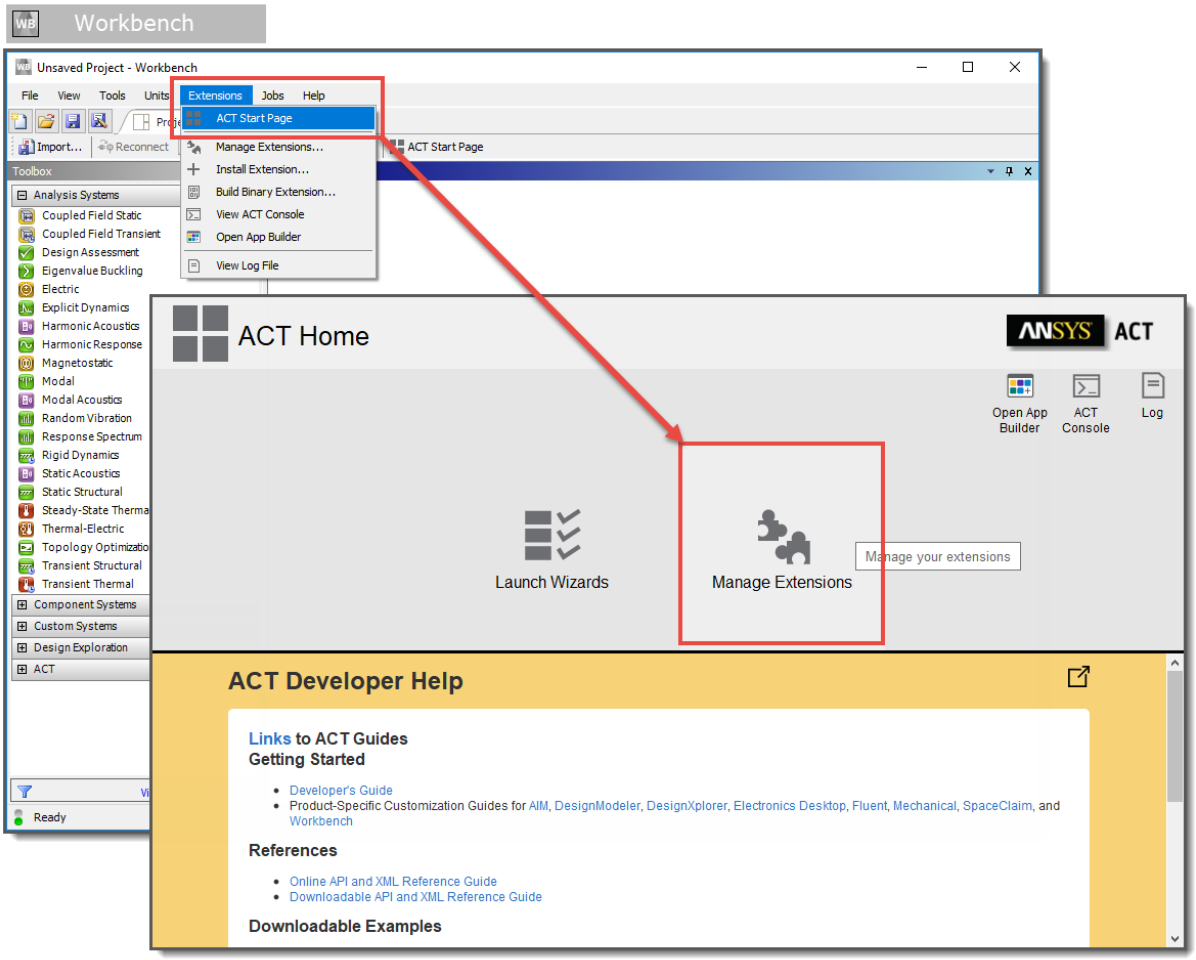
Setting Up the AdditivePrintTransfer Extension

This is a one-time-only step.

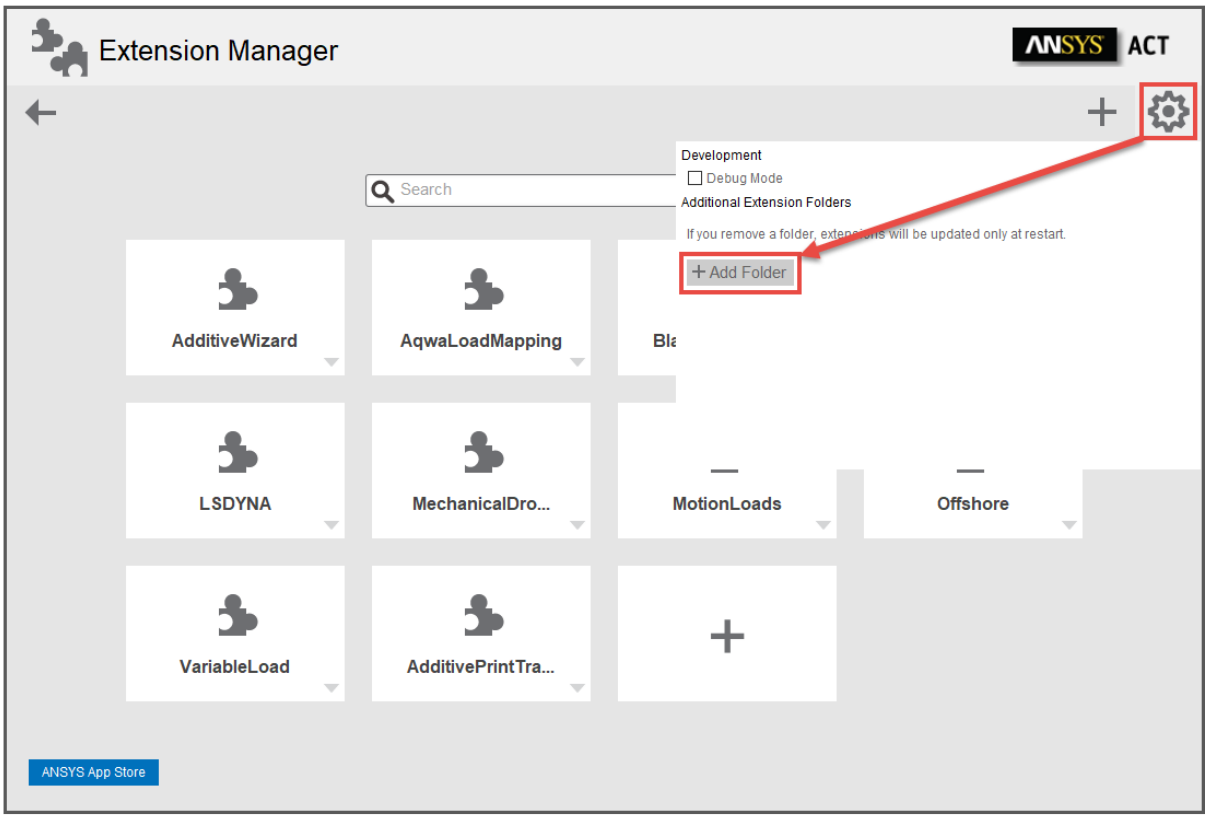
1. Download the AdditivePrintTransfer.zip file from this site (where you downloaded this document).
2. Extract the zipped files. You should have the following folder structure:

```
AdditivePrintTransfer
      doc
      src
```

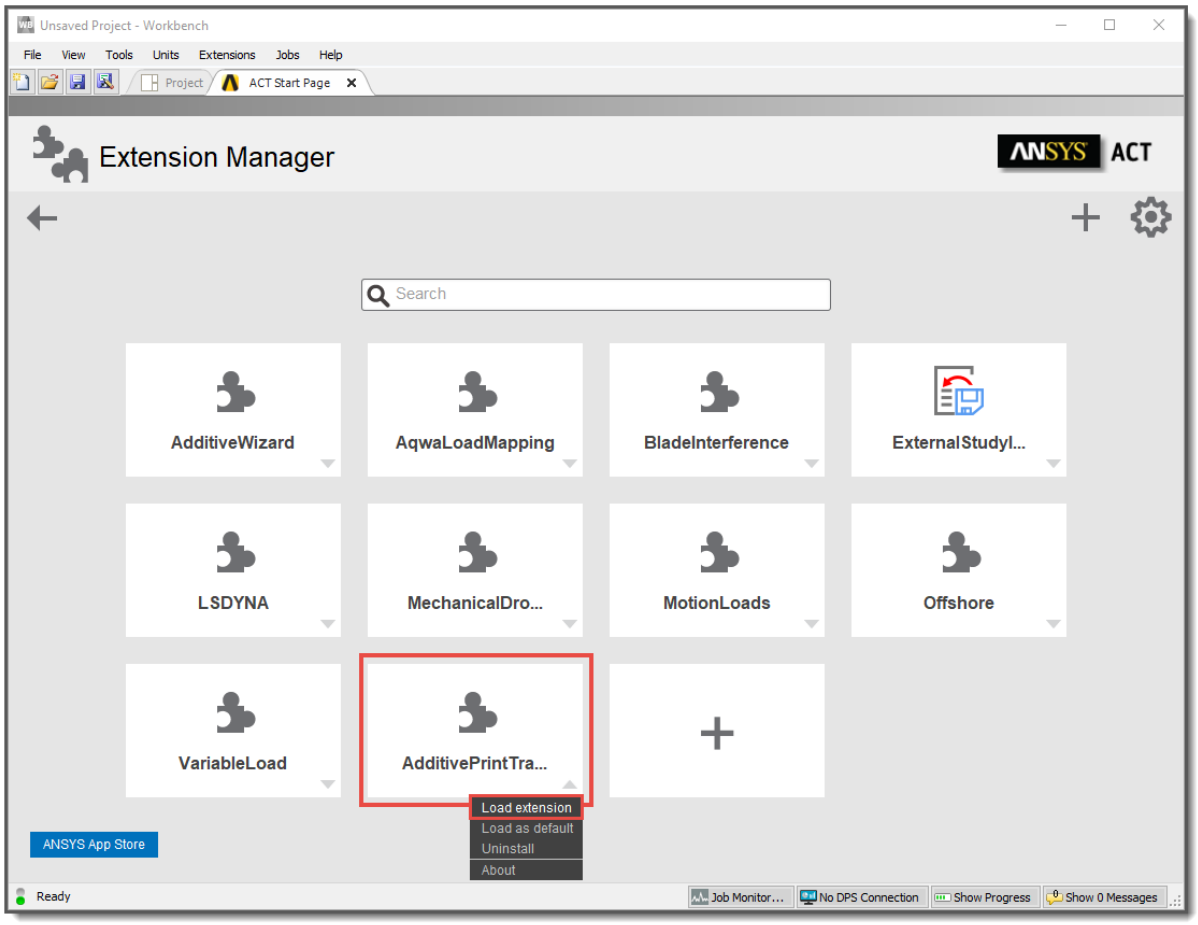
3. Open Workbench 2020 R2.
4. Click **Extensions** > **ACT Start Page** > **Manage Extensions**



5. Click the **gear icon** (Manage your settings) in the upper, right corner to add a new folder for the extension. Click **Add Folder**. Navigate to the *"src" folder* location of the AdditivePrintTransfer files. Click **Open**.



6. You will see the new extension in the Extension Manager. From the AdditivePrintTransfer box drop-down, click **Load extension**. We recommend you also then click **Load as default** from the drop-down so that the extension will be loaded whenever you open Workbench. (Extensions that load by default appear as green boxes in the Extension Manager.)

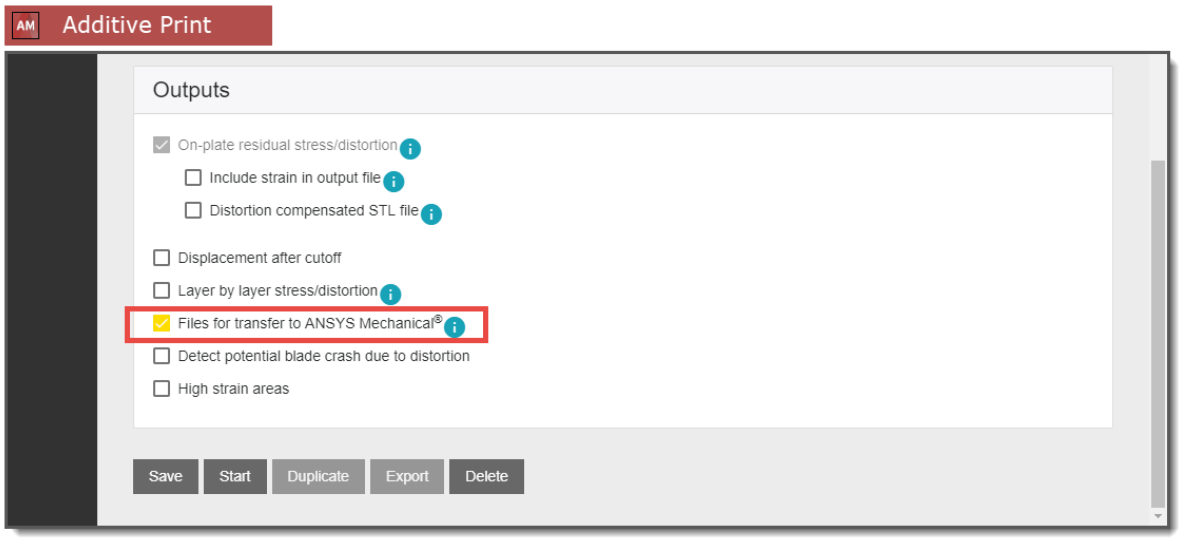


Obtaining Cutoff Results

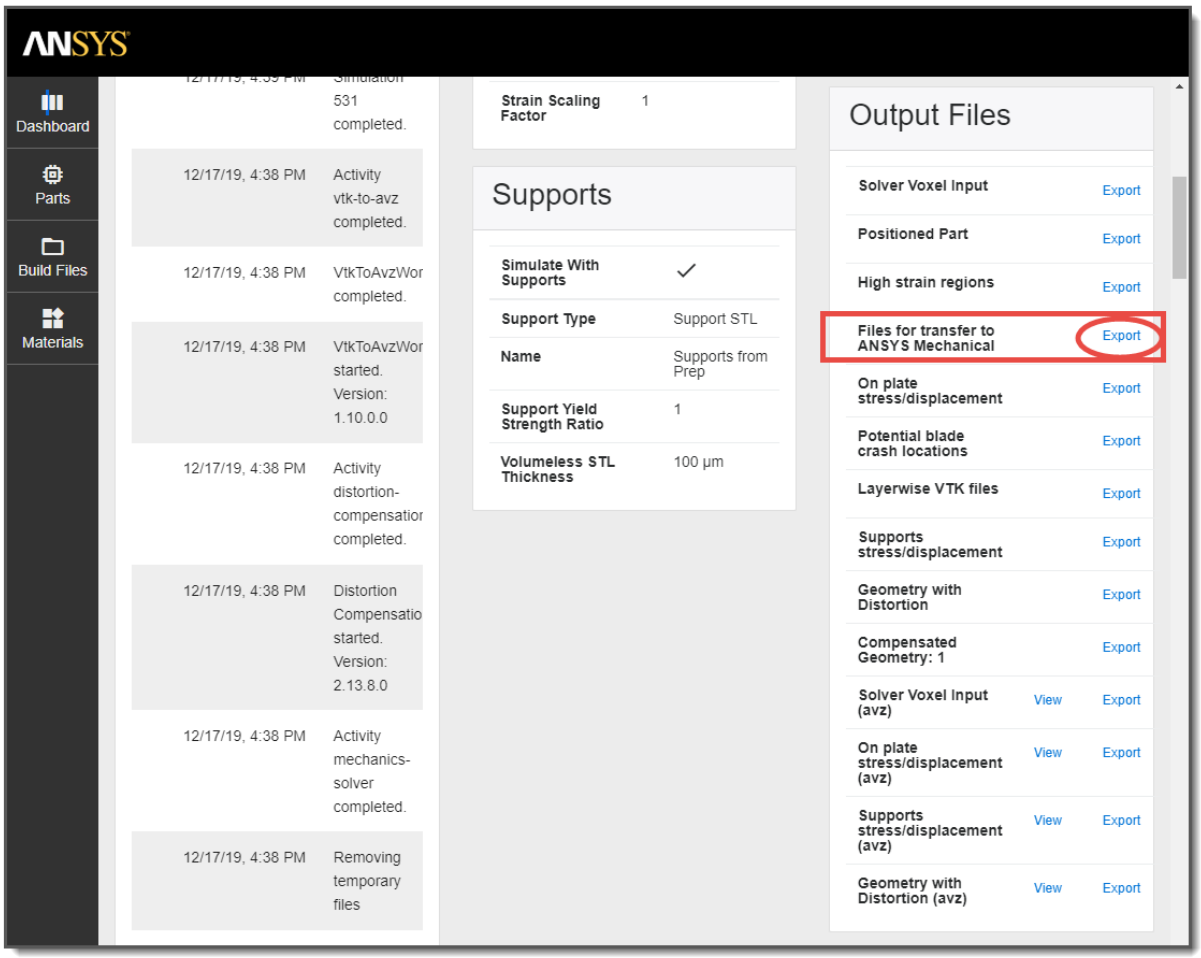
Perform the following steps whenever you want to use Workbench Additive to study after-cutoff results from a simulation performed in Additive Print.

1. Set up a simulation in Additive Print 2020 R2. (This entire procedure will not work with simulations from previous versions of Additive Print because the required output files were not written.)

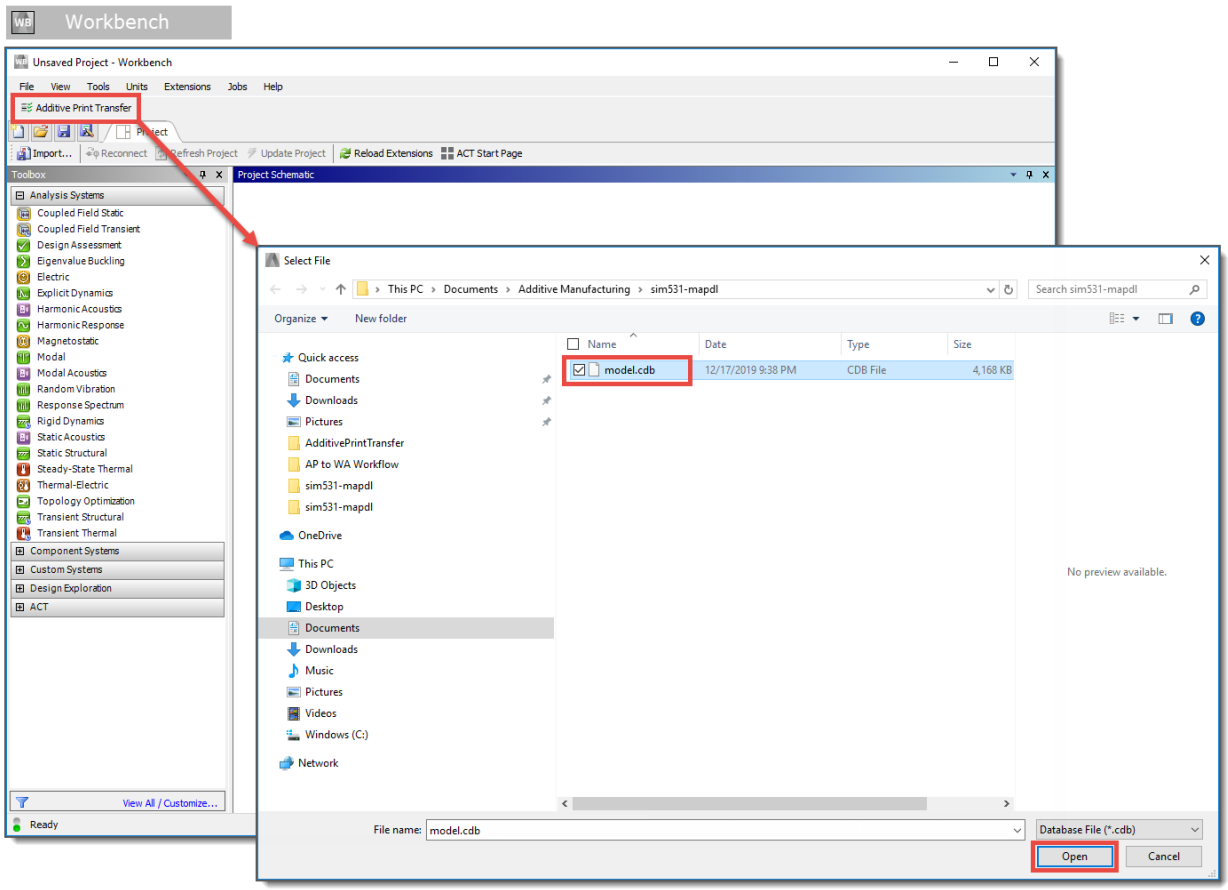
2. When selecting output options, select the *Files for Transfer to ANSYS Mechanical* option.



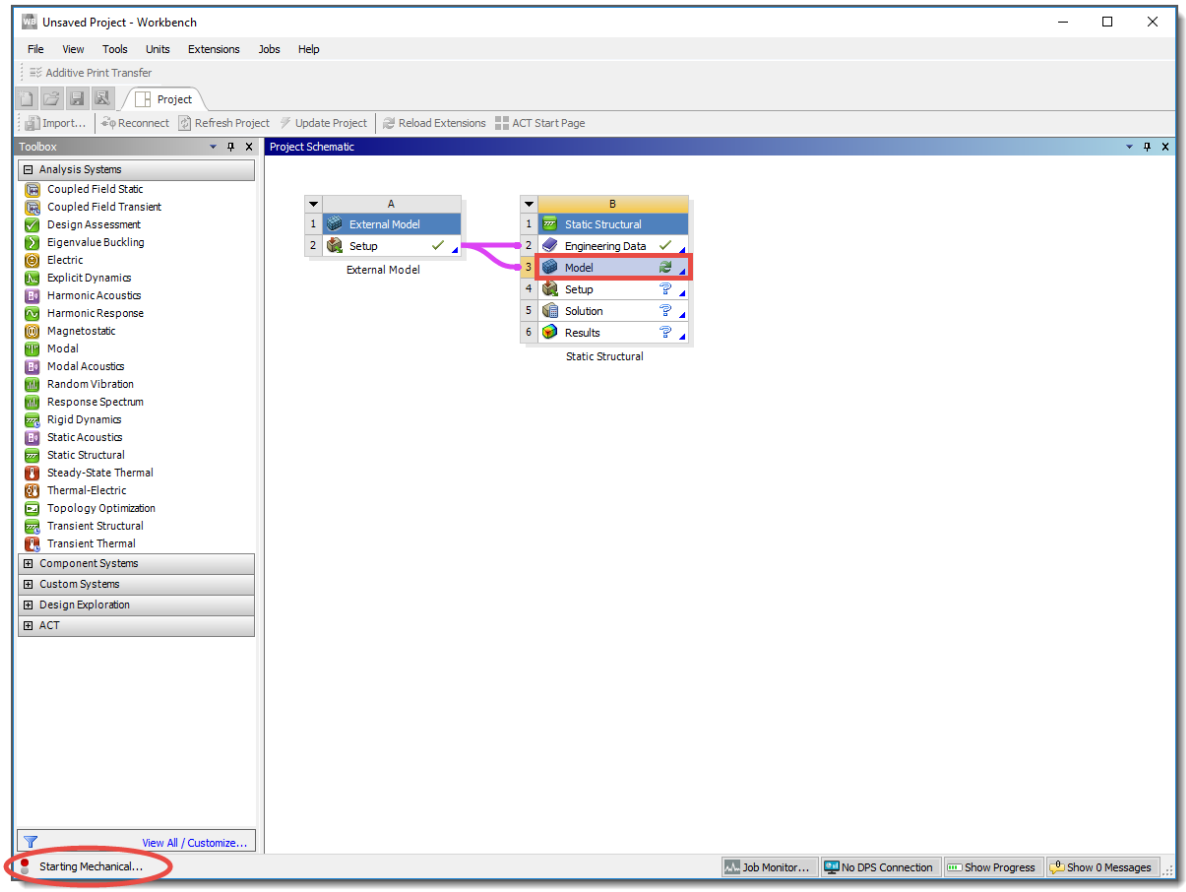
3. Run the simulation as normal.
4. Open the completed simulation. Export the *Files for Transfer to ANSYS Mechanical* output file, which is named `sim#-mapdl.zip`, to a directory location on your computer. Keep this path name short, as later, in Mechanical, there is a 128-character limit when specifying the file path name.



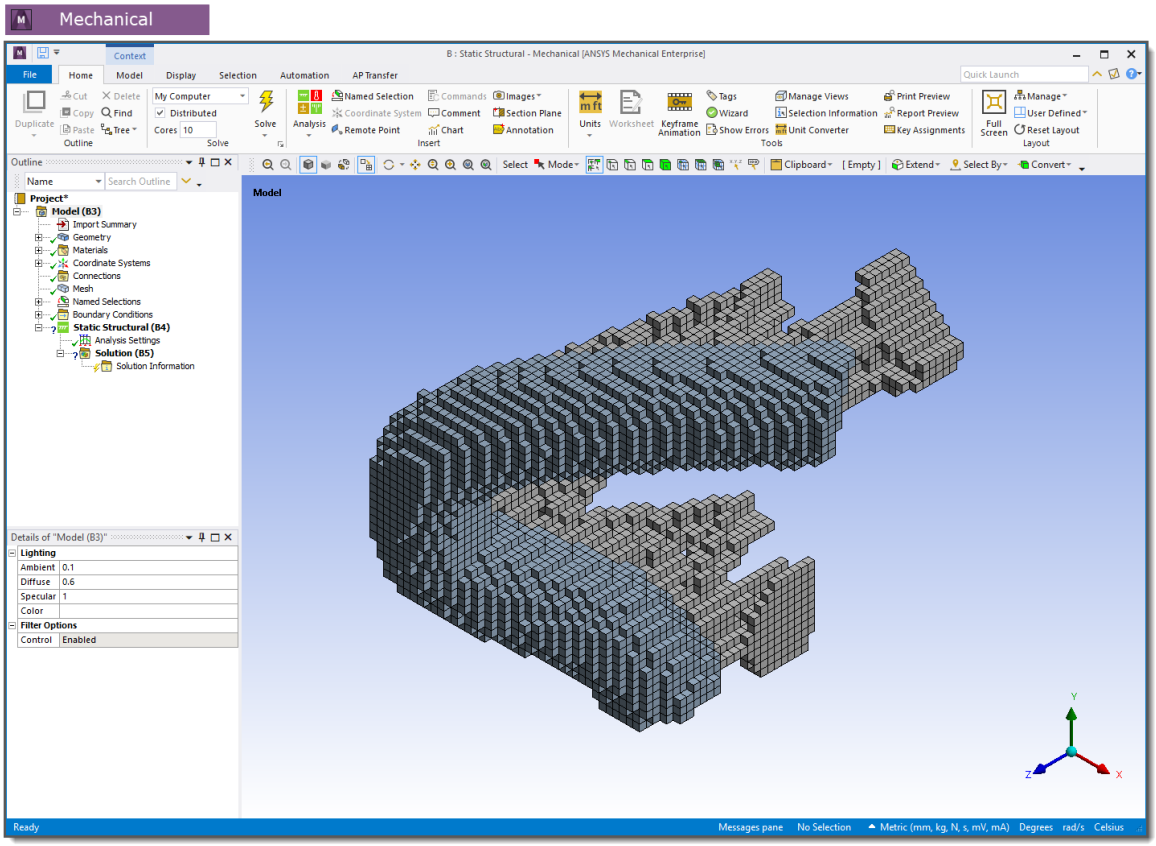
5. Unzip the `sim#-mapdl.zip` file
6. Open Workbench 2020 R2.
7. Load the AdditivePrintTransfer extension (if not loaded by default). **Extensions** > **Manage Extensions** > Check **AdditivePrintTransfer** > **Close**.
8. Click on **Additive Print Transfer** button. This brings up the file manager. Navigate to the directory location of your Additive Print output files and click **`model.cdb`** > **Open**.



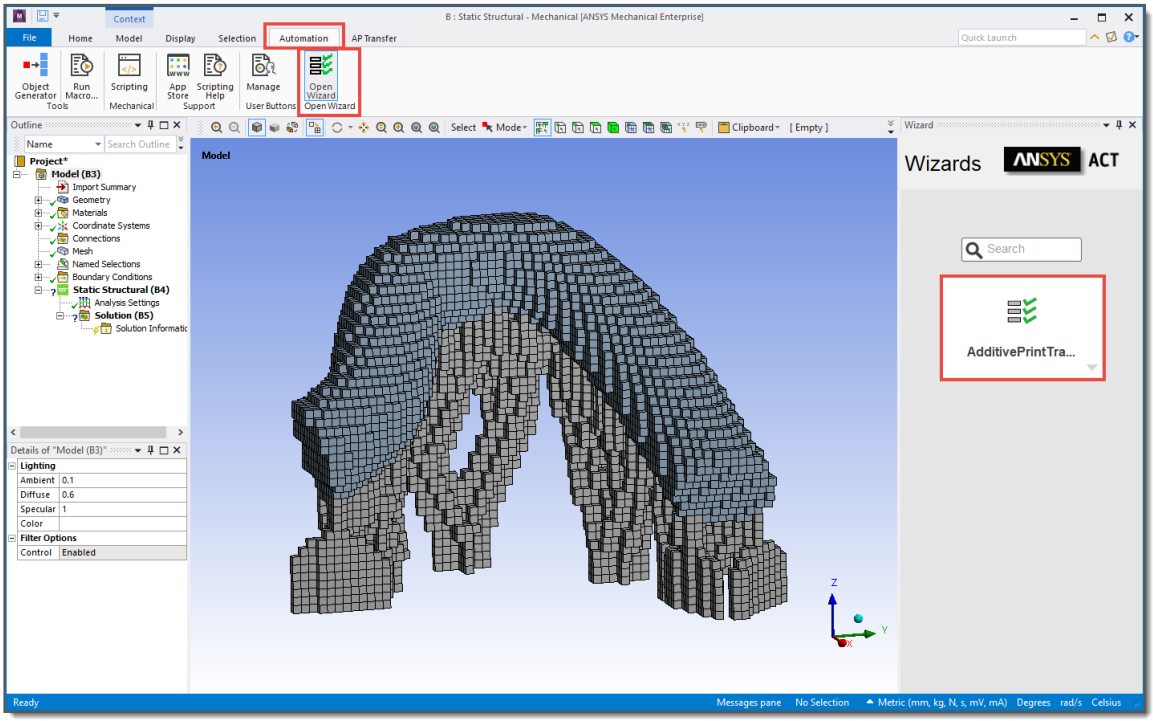
9. Double-click the **Model** cell in the Static Structural system to open Mechanical. It usually takes a minute or two for the application to open. Look for the “Starting Mechanical” message in the status bar for confirmation that the application is opening.



Once Mechanical is open and your model is loaded, feel free to move the model around with your mouse. Click on an axis arrow in the triad to align the model in any direction.



10. Click **Automation** tab > **Open Wizard** > **AdditivePrintTransfer** to open a wizard that guides you through the process.



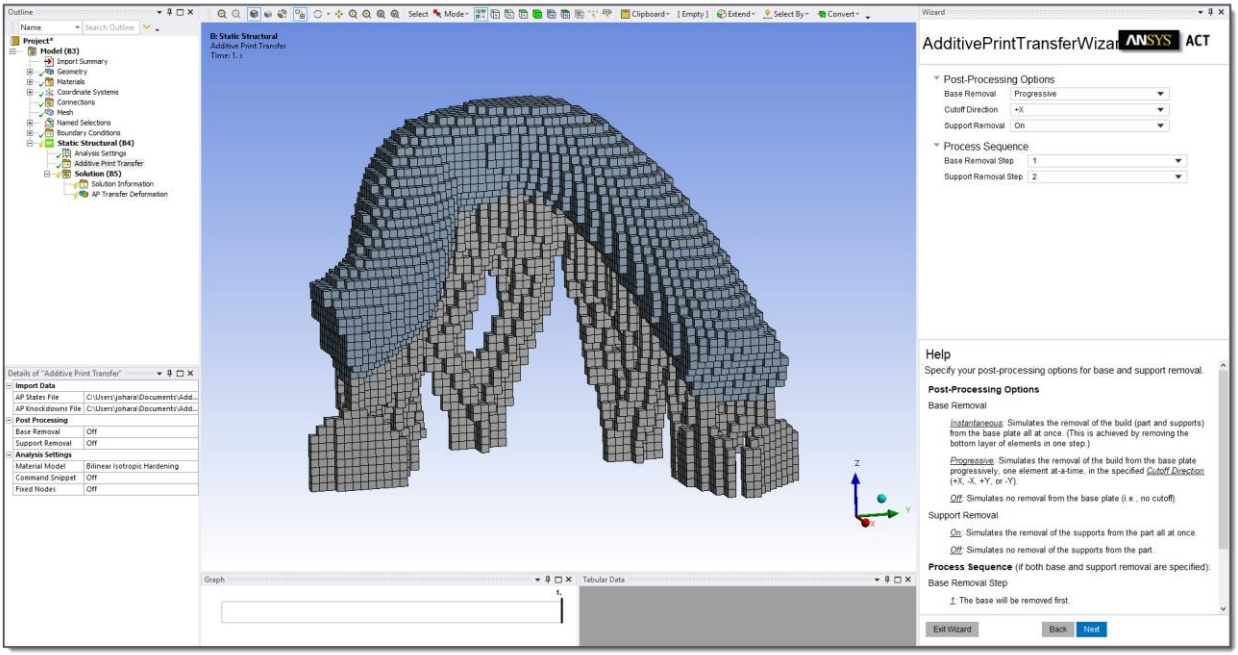
11. To import the remaining data from the Additive Print simulation, identify the file locations of the states file, the knockdown factors file, and the displacements file. The file locations default to the same folder that you specified for the `.cdb` file so you shouldn't have to change anything. Click **Next**.
    - **States.ist** – A file containing Additive Print's on-plate end-state information (elastic strains, plastic strains, and accumulated equivalent plastic strain). This will provide the initial stress state for subsequent processing in Mechanical.
    - **Knockdowns.ist** – A file containing element (voxel) density information.
    - **Displacements.csv** – A file containing Additive Print's on-plate end-state displacements.
12. Specify your post-processing options for base and support removal.

    Base Removal
    - **Instantaneous** will simulate the removal of the build (part and supports) from the base plate all at once. (This is achieved by removing the bottom layer of elements in one step.)
    - **Progressive** will simulate the removal of the build from the base plate progressively, one element at-a-time, in the specified direction (+X, -X, +Y, or -Y) and with a specified number of cutoff steps.
    - **Off** will simulate no removal from the base plate (i.e., no cutoff).

    Support Removal
    - **On** will simulate the removal of the supports from the part all at once.
    - **Off** will simulate no removal of the supports from the part.

    If both removal options are specified, process *sequence* options appear where you will specify the removal sequence for both steps (1 for first, 2 for second).

    Click **Next**.

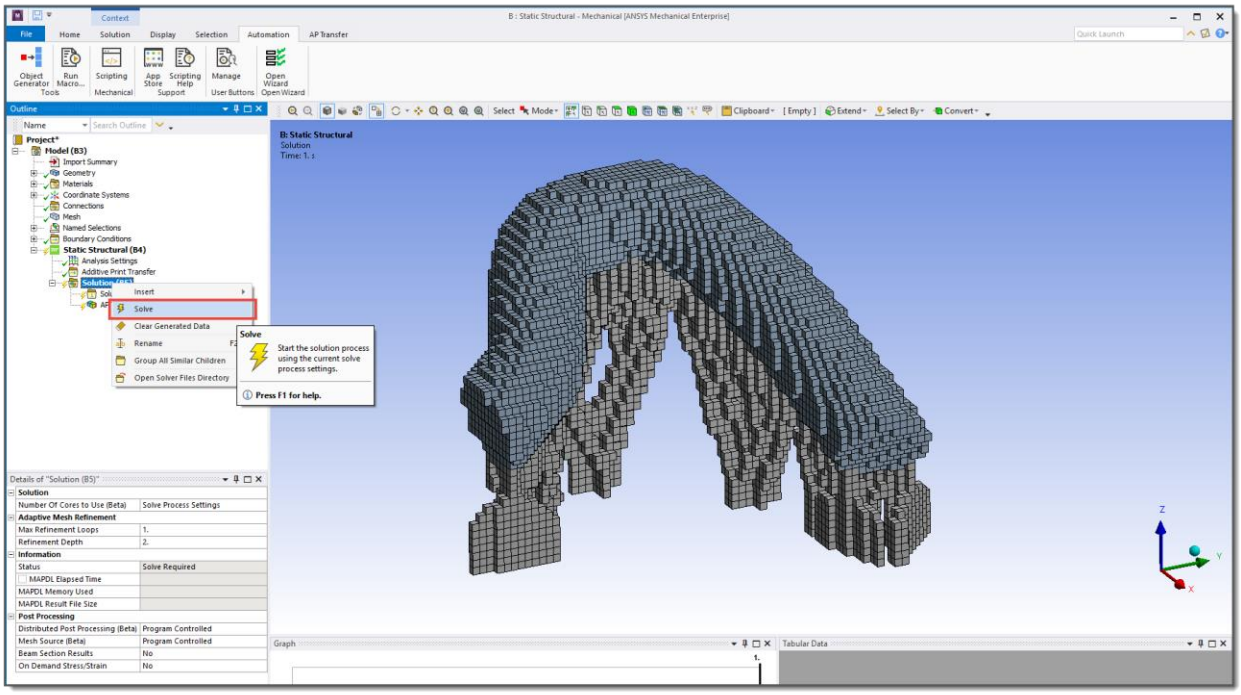


13. The option to fix nodes is available if you specified a removal step. We suggest you use the default **Off** option at first. If you run into convergence problems or a simulation failure, then turn this option to **On** to constrain the part at three nodes to prevent rigid body motion.

    To constrain the part (i.e., Fix Nodes for Cutoff = On), choose nodes that are unimportant to your results. Perhaps spread them out over the bottom of the part but the nodes must not be co-linear. Your node selection will affect displacement results but not stresses or strains. Do not select nodes on any supports if you specified to remove supports.

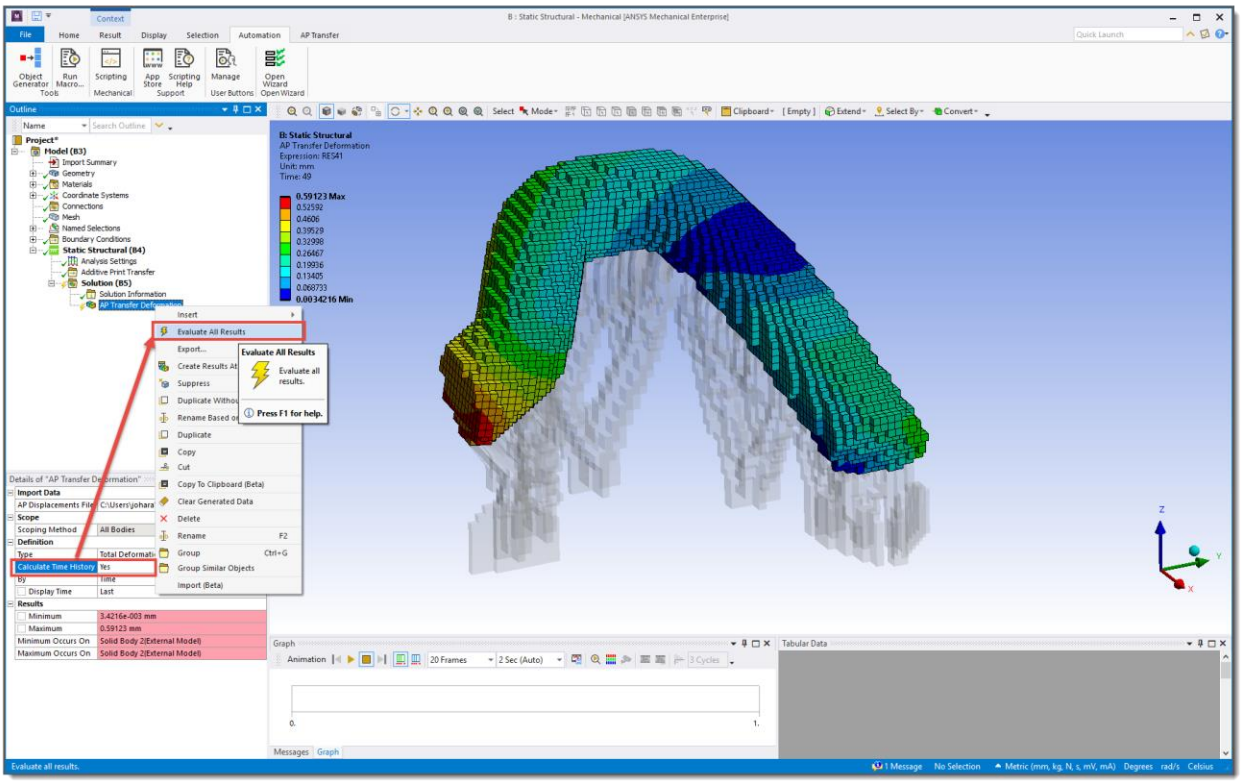
    First switch the mode of selection to Node.  You may need to hide the base plate to select nodes on the bottom of the part. To do so, click on the base plate geometric entity under Model, Geometry in the project tree and hit the F9 key. F9 toggles the geometric entity visibility on and off. Then select three nodes on the part. Click Apply after each selection. Constrained Node 1 is fixed in UX, UY, and UZ. Constrained Node 2 is fixed in UY and UZ. Constrained Node 3 is fixed in UZ.

14. Click **Finish** to end the wizard session. Close the wizard box by clicking the X in the upper, right corner of the wizard window.
15. Right-click **Solution** in the project tree and click **Solve**. You may see a warning message about not enough constraints to prevent rigid body motion, but the solution will continue. Keep going. Watch the status bar at the bottom, left corner to observe status updates.

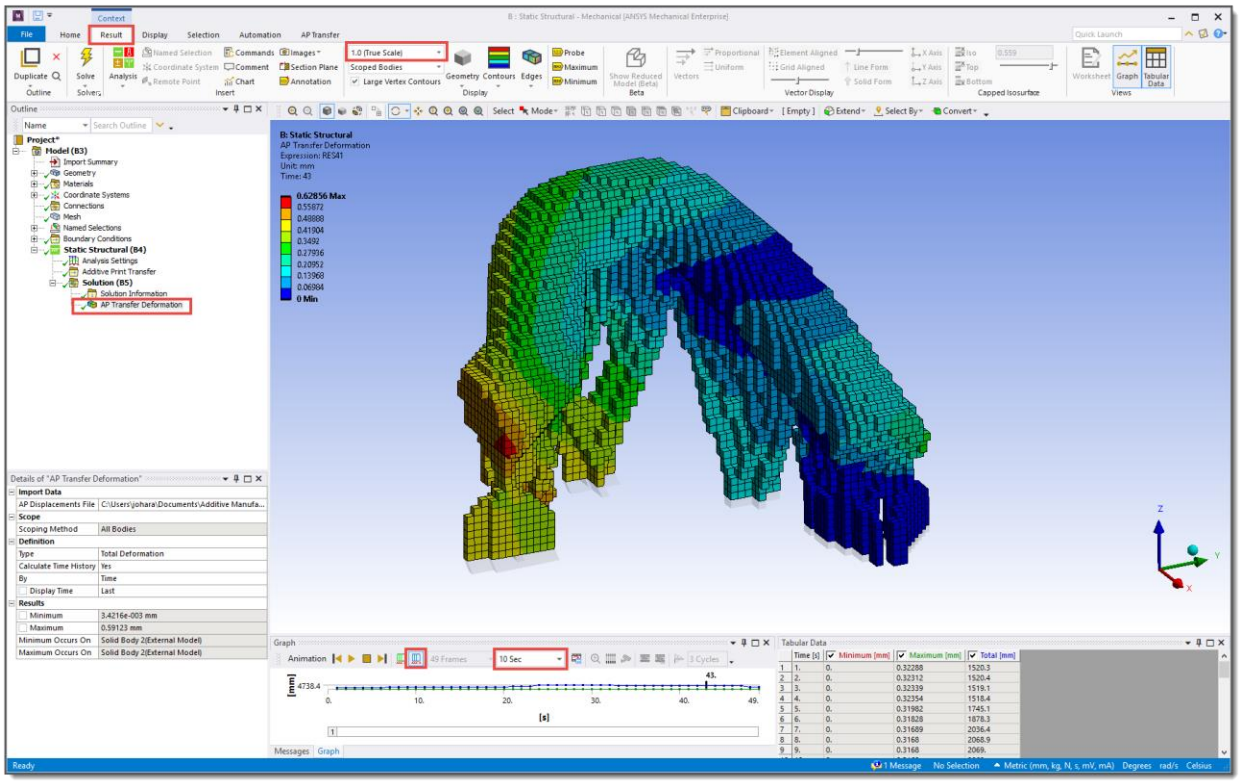


16. You will know when the solution is complete when you see little green checkmark icons next to the solution objects in the project tree. To view deformation results, click in **AP Transfer Deformation** in the project tree. In the Details view of AM Deformation, you will see that Calculate Time History is No by default. Set **Calculate Time History** to **Yes** to process the results for every step in the simulation. Right-click AP Transfer Deformation and select **Evaluate All Results**. This will take longer but is necessary if

you want to see animation of the simulation results.



17. To view an animation of your cutoff results, click **AP Transfer Deformation** and set the animation controls on the bottom as shown below. Finally, click the **Result** tab in the ribbon and change to **True Scale**. (The deformation animation is shown exaggerated by default.) Click the play triangle.

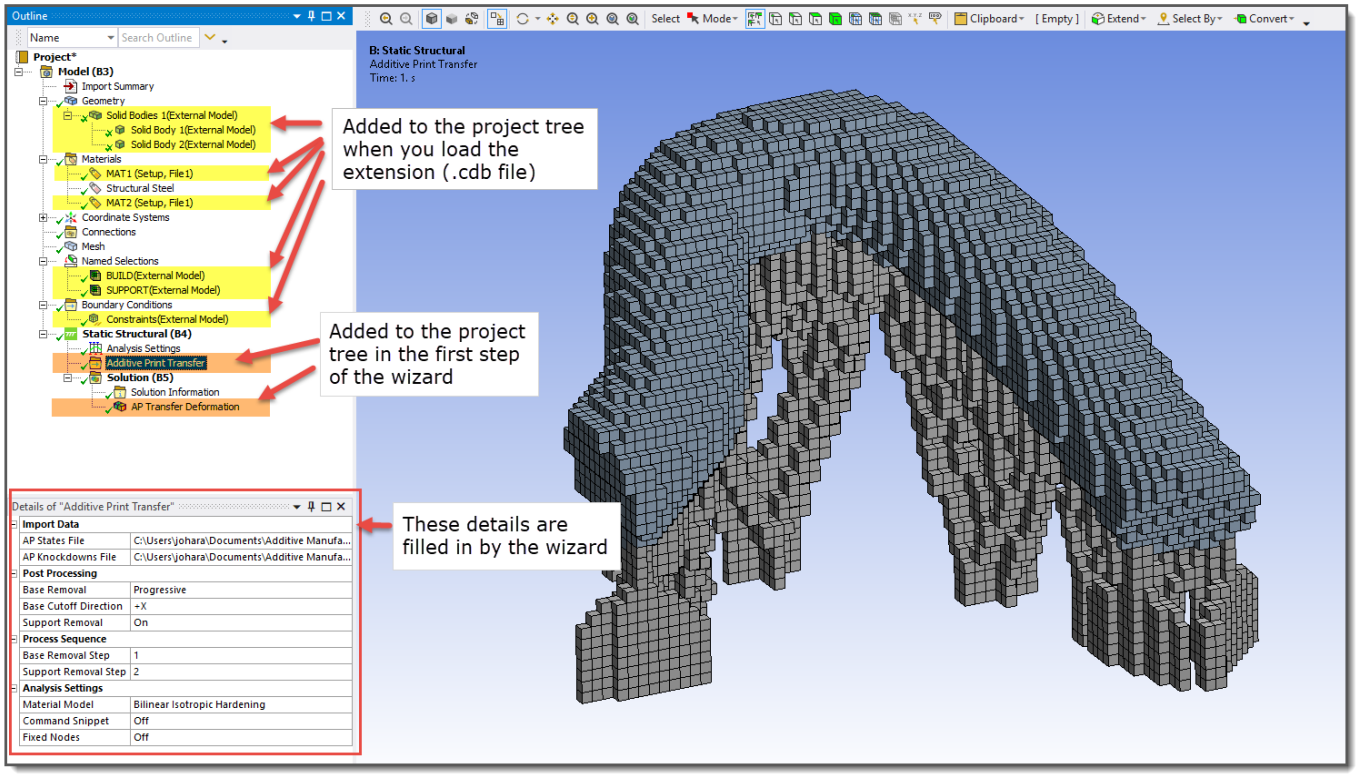


The AP Transfer Deformation in the project tree shows total deformation results that are the sum of the on-plate displacements (from Additive Print) and the cutoff displacements (from Workbench Additive). These are the results most people are interested in. (If you insert a standard Total Deformation tracker into the tree, your results will show cutoff displacements only. It will not include the displacements at the end of the build while the part is still attached to the baseplate.)

An Alternative to the Wizard

Using the wizard as described in the above steps is just one way to obtain cutoff results using the AdditivePrintTransfer extension. Sometimes it is beneficial to know what is going on behind the scenes of the wizard, as this leads to greater understanding of the powerful Mechanical application. Every step of the wizard

adds or edits items in the project tree, as shown here.



It is quite easy to manually configure the settings to perform the AdditivePrintTransfer function. You may have noticed the AP Transfer tab in the ribbon. This is automatically added by the extension. Click on the **AP Transfer** tab and then click the **Additive Print Transfer** tool to add the object into the project tree. Configure all the settings as desired in the **Details** view. An AP Transfer Deformation object is added into the tree when you click the Additive Print Transfer tool. It has total deformation specified by default. Click the AP Transfer Deformation tool to add new deformation objects into the tree if you want to solve for a directional type of deformation (X, Y,

or Z). Solve and review results as described in the above steps.