Netfabb Local Simulation

Examples Manual

Autodesk Inc.[1]
Software Solutions for 3D Printing

Version 19.1

[1] Email:projectpansupport@autodesk.com

THIS PUBLICATION AND THE INFORMATION CONTAINED HEREIN IS MADE AVAILABLE BY AUTODESK, INC. AS IS. AUTODESK, INC. DISCLAIMS ALL WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE REGARDING THESE MATERIALS. AUTODESK WILL NOT BE LIABLE IN ANY MANNER FOR THE RESULTS OBTAINED THROUGH USE OF THE PUBLICATION.

Contents

Introduction

This manual contains examples of using Netfabb Simulation to simulate additive manufacturing processes.

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page. The folder name corresponds with each example number.

Example 1

Thermo-Mechanical Process Parameter File Generation

1.1 Problem Description

This example illustrates how to generate a thermo-mechanical Process Parameter file, known as a PRM file. A PRM simulation models a small amount of material to determine how a certain material will thermo-mechanically respond to a certain set of processing parameters. This information gets encoded in the PRM file, which is read by subsequent part-level analyses for builds using the same material and processing parameters.

Instructions on how to produce a thermal prm file to investigate lack of fusion and hotspot behavior are given in Example 14.

This example will also guide the user through how to produce post-process time-temperature, and time-displacement data files for selected points, which can be used to plot thermal or displacement results.

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

In order to run Part-Level Powder-Bed analysis in Netfabb Simulation , a process parameter (.prm) file must first be generated. The .prm file links the small scale moving-source analysis to the full Part-Level analysis. This is illustrated in Figure 1.1.

Here, a process parameter file is generated for Inconel 625 using the following set of parameters:

- Power: 150 W
- Laser spot size: 0.15 mm
- Scan speed: 600 mm/s
- Layer thickness: 0.04 mm
- Hatch spacing: 0.15 mm
- Recoater time: 20 s
- Initial angle rotation: 11.5 degrees
- Interlayer hatch angle rotation: 67 degrees

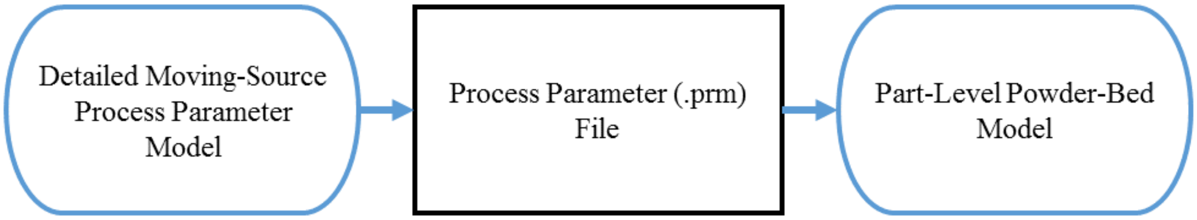

Detailed Moving-Source Process Parameter Model
Process Parameter (.prm) File
Part-Level Powder-Bed Model

Figure 1.1: Relationship between the fine scale and Part-Level analyses.

The parameters are entered into the *LSRP card. The *GTAB card enables PRM file output and specifies the name of the process parameter file. The flow of the analysis is shown in Figure 1.2.

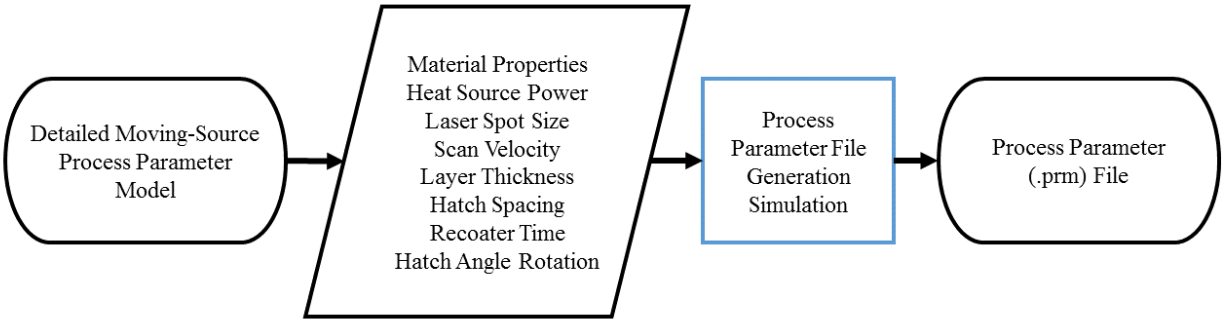


Figure 1.2: Flowchart for generating .prm files.

A time incremental thermal analysis is performed first to compute the temperature history of the part followed by a time incremental mechanical analysis. The .prm file is filed out for several different section thicknesses and temperatures. The thickness of a section is controlled by using the 10th input of the *LSRP card and the temperature is controlled by using the *INIT card. Once the full table is filled out in the .prm file, the file can be input along with geometric information for the Part-Level analysis as illustrated in Figure 1.3. Part-Level analyses are demonstrated in later examples in this manual.

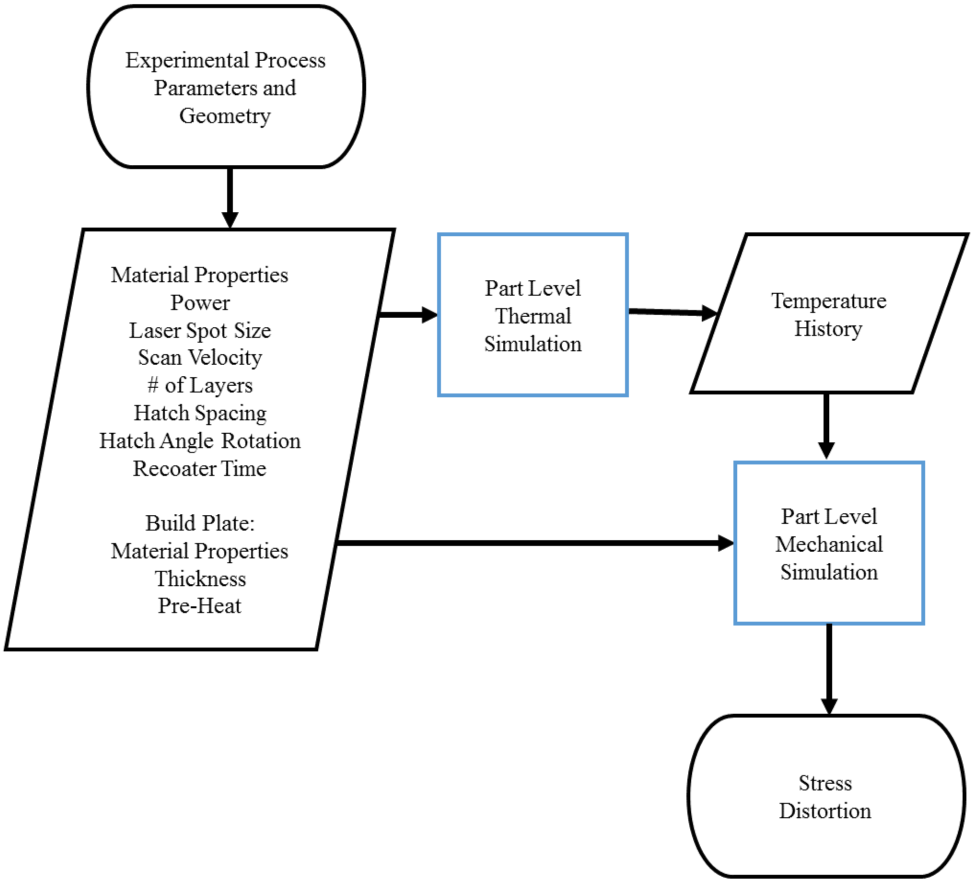


Figure 1.3: Flow chart from fine scale to part scale thermo-mechanical simulations

1.2 Running Netfabb Simulation

To run the models, from a command line run:

```
$ prm_gen 01_thermal.in 01_mechanical.in > prmgen.out
```

Users can check the progress of the simulation by viewing the log file, which is recorded to the prmgen.out file.

This will run each combination of temperature and thickness in order

1.3 Results

The result of the analysis will be a single process parameter (.prm) file. The file will be read into succeeding Powder-Bed Part-Level analyses.

1.4 Post processing

A tool for producing temporal results, timex, is included in the installation. This program uses an input text file with the following entries:

```
*INPU
a1 = input-file-name (without *.in extension)
*PNTS
i1 = Number of points to probe
r11, r12, r13 = X, Y, Z coordinates, point 1
r21, r22, r23  = X, Y, Z coordinates, point 2
...
```

Two timex input files are included, timex-temp.txt and timex-disp.txt, which probe the thermal and mechanical results at several locations, respectively.

To produce a temperature history for selected points, from the command run:

```
$ timex timex-temp.txt
```

The resulting comma separated text file is called timex_prmgen_thermal.txt. It has the format: Time (s), Temp at Point 1, Temp at Point 2, Temp at Point 3, ...

View the timex_prmgen_thermal.txt file in the text editor of your choice. Note that for locations which are in the deposition region, temperatures are 0 until the associated element has been activated. This data is easy to plot in any spreadsheet software or programming environment.

To produce a displacement history for selected points, from the command run:

```
$ timex timex-disp.txt
```

The resulting comma separated text file is called timex_prmgen_mechanical.txt. It has the format: Time (s), Point 1 Displacement Magnitude, Point 1 X Displacement, Point 1 Y Displacement, Point 1 Z Displacement, Point 2 ....

Open up the timex_prmgen_mechanical.txt file. Note similarly to the thermal results, all displacements are set to 0 before the element has been activated.

Example 2

Part Scale Modeling

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

2.1 Problem Description

A generic geometry of Inconel®625 is built in a powder bed system and simulated. The layer height is 0.04 mm. The part geometry is imported in the analysis through an STL file, and it is automatically meshed within Netfabb Simulation . The substrate is assumed to be 24 mm thick. The actual build plate is planned to have 5 similar geometries on it. Here, a simplified analysis is performed on just 1 of the geometries. The *PBDL card is used to add the dwell time for the deposition of the geometries that are not included in the analysis. The *PBIS card insulates the side of the small substrate in the analysis, simulating the effect of having other builds on the build plate nearby. The *PBSS card constrains the sides of the small substrate in the analysis, mimicking the effect of being attached to the larger build plate. The build plate has an initial temperature of 100°C, which is modeled using *INIT. The resulting mesh is illustrated in Figure 2.1.

A time incremental thermal analysis is performed first to compute the temperature history of the part. Layers are activated in groups, and additional time increments are used to model heat conduction into the part. The thermal analysis includes only the part and substrate. Heat loss into the powder is modeled as convection with a value of 25.d-6 W/((mm^2)°C) using the *CONV option.

A time incremental mechanical analysis is performed after the thermal analysis is completed. Similarly to the thermal analysis, layers are activated in groups using *PBPA and the computed temperature distribution from the mechanical analysis is used to compute deformation due to the thermal expansion. The input process parameter file (Inconel625_generic.prm) was generated in Example 1 of this manual.

After the thermo-mechanical simulation has been completed, the `distort_stl` post-processing program will be used to produced both a warped STL which shows the predicted displacements and a compensated STL, which if printed, should mitigate much of the distortion of the original geometry, getting the part closer to the desired shape.

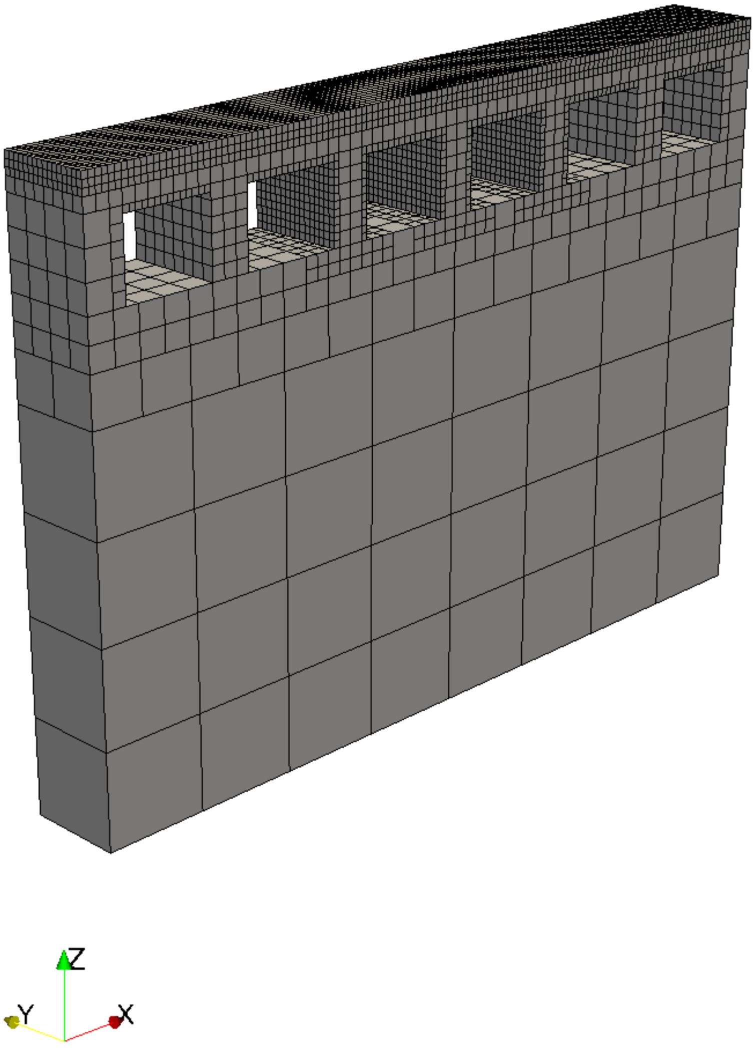


Figure 2.1: Auto-generated finite element mesh

2.2 Running Netfabb Simulation

2.2.1 Thermal Analysis

To run the model, from a command line run:

```
$ pan -b 02_thermal
```

The **-b** option runs the solver in background mode, which automatically overwrites any previous results, and directs output to a an output file of the format `input-file-name.out`.

The analysis progress is written on file `02_thermal.out`. To check progress in a linux environment run:

```
$ tail 02_thermal.out
```

To check progress in a windows command line environment run:

```
$ type 02_thermal.out
```

After the analysis completes, the last few lines of the output file `02_thermal.out` should be similar to the following:

```
Increment end
 inc =       35 time =   4249.1602     iter =   1 eps =  0.23990E+03
 inc =       35 time =   4249.1602     iter =   2 eps =  0.42313E-12
Finished writing file results/02_thermal_35.case
Writing record:            2 , time:    4249.16015625000
Increment end

Mesh preview volume  =    761.062500000000
Activated volume     =    761.062500000000
Activated percentage =    100.000000000000

Finished writing file ./02_thermal.case

Analysis completed

************************
         1 Warning
************************

CPU wall=   12.25391     CPU total=   40.10786
Peak RAM used for this process = 97384 kB

END Netfabb Simulation Solver
```

Actual CPU times will differ from system to system.

Note that a warning was produced during this simulation. All warnings from this simulation are written to an additional log file `02_thermal_warning.txt`. Open or tail the file to see the warning:

```
*** WARNING:            1
STL is too small.  Check units.
```

This warning lets the user know that the volume of the STL is very small, and may not be in the default mm units. This is merely a small test geometry, and the warning may be ignored.

2.2.2 Quasi-Static Mechanical Analysis

Run the analysis from the command line:

```
$ pan -b 02_mechanical
```

The analysis progress is written on file `02_mechanical.out`. To check progress run:

```
$ tail 02_mechanical.out
```

or in Windows:

```
$ type 02_mechanical.out
```

After the analysis completes, the last few lines of the output file `02_mechanical.out` should be similar to the following:

```
 ----------------------------------
  Substrate removal time increment
 ----------------------------------
  inc =       37 time =   7069.1602     iter =   1 eps =  0.53336E+04
  inc =       37 time =   7069.1602     iter =   2 eps =  0.10623E-08
 Finished writing file results/02_mechanical_37.case
 Increment end

 ------------------------------------------------------
  Total number of equilibrium iterations:           73

 Mesh preview volume  =    761.062500000000
 Activated volume     =    761.062500000000
 Activated percentage =    100.000000000000

 Finished writing file ./02_mechanical.case

 Analysis completed

 ************************
           1 Warning
 ************************

 CPU wall=   22.41406     CPU total=   76.72129
 Peak RAM used for this process = 335940 kB

 END Netfabb Simulation Solver
```

Actual CPU times will differ.

2.3 Results

Results may be imported and viewed in Paraview or the Simulation Utility for Netfabb. Figures 2.2 shows the computed final distortion after substrate release.

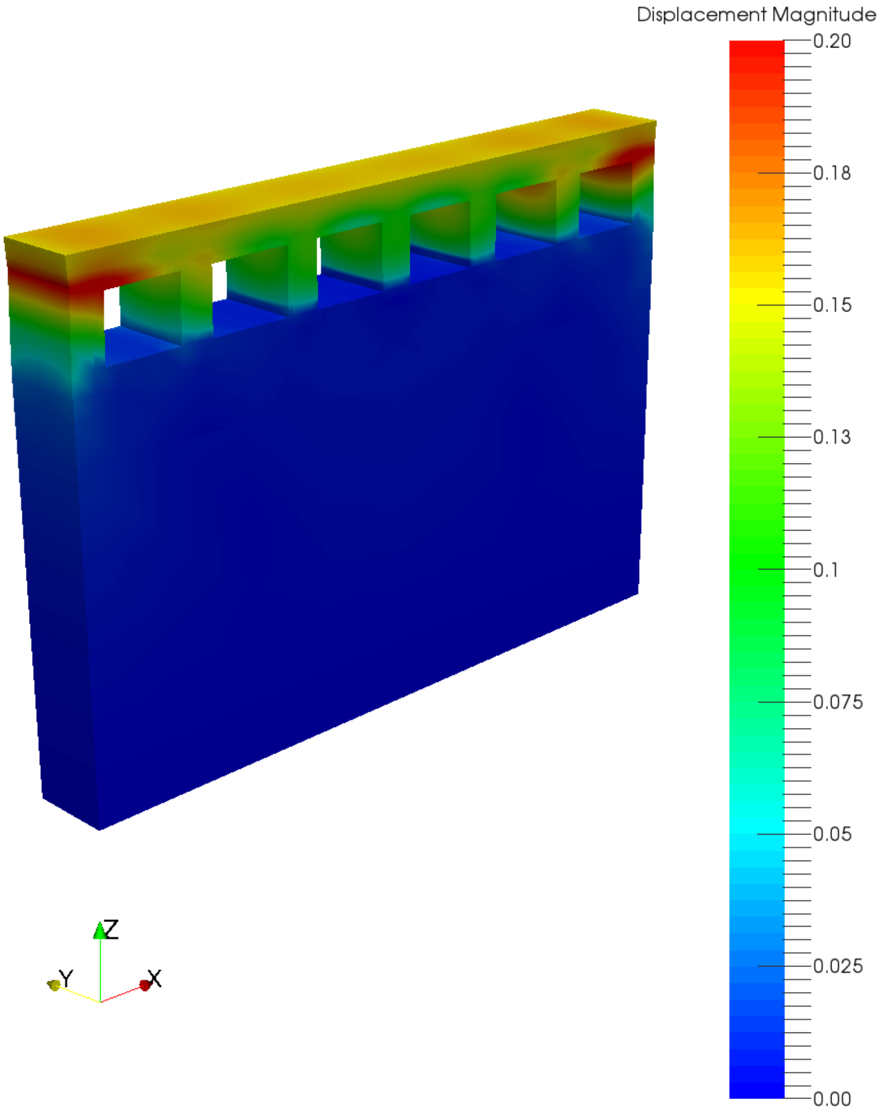


Figure 2.2: Final distortion.

2.4 Producing distorted and compensated STL files from the simulation results

After the thermo-mechanical simulation is completed, the mechanical results can be used to output warped and compensated STL files. Warped STLs can be used for post-process analysis, to ensure assembly fit or other dimensional checks. To produce a warped STL, use the included program `distort_stl`.

```
$ distort_stl warp.txt
```

By default the program uses the distortion results after cool down but before removing the part from the build plate. The resulting distorted STL, 02_mechanical_warp.STL, is shown in Figure 2.3.

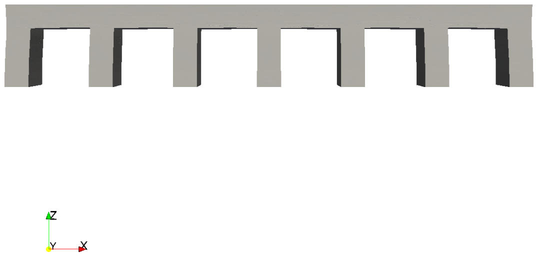


Figure 2.3: Warped STL

A compensated STL takes the prediction distortion results, inverts them, and applies them to the original geometry. This produces a geometry which should distort *into* the desired shape. To produce the compensated geometry, use the `distort_stl` program again:

```
$ distort_stl comp.txt
```

The resulting compensated STL, 02_mechanical_compensated.STL, is shown in Figure 2.4.

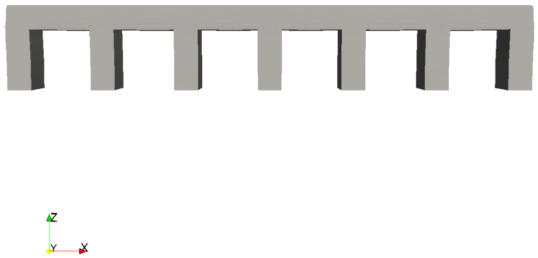


Figure 2.4: Compensated STL

Example 3

Advanced Part Scale Modeling

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

3.1 Problem Description

For this example showcasing some advanced part-scale modeling option, a sample geometry simulates the powder bed construction of an Inconel®625 part. The layer height is 0.04mm. The part geometry is imported in the analysis through an STL file, and it is automatically meshed within Netfabb Simulation . The buildplate is modeled to be 25mm thick and 60mm x 60mm in area as defined in the *SBDM option. The substrate is fixed to a circular rod defined using the *FIXC card. A controlled temperature of 200°C is applied to the build plate using *PBSH. The resulting mesh is illustrated in

Figures 3.1.

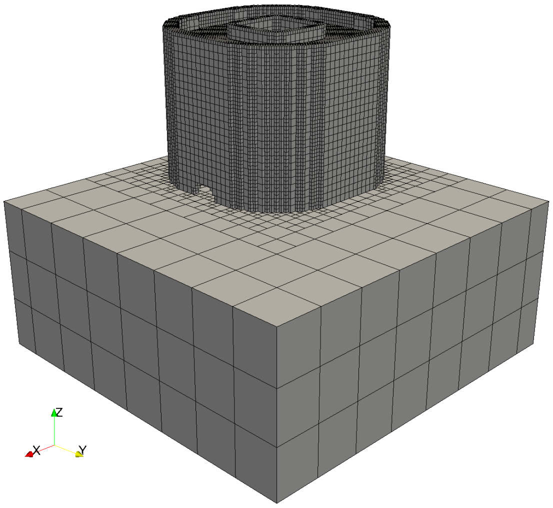


Figure 3.1: Mesh.

A time incremental thermal analysis is performed first to compute the temperature history of the part. Layers are activated in groups using *PBPA, and additional time increments are used to model heat conduction into the part. The thermal analysis includes only the part and substrate.

Heat loss into the powder is modeled as convection with a value of 25.d-6 W/((mm^2)°C) using *CONV.

A time incremental mechanical analysis is performed after the thermal analysis is completed. Similarly to the thermal analysis, layers are activated in groups and the computed temperature distribution from the mechanical analysis is used to compute deformation due to the thermal expansion. The input process parameter file (Inconel625_generic.prm) was generated in Example 1 of this manual. The *WRTU option is used to output two point cloud files, before and after removal of the part from the build plate.

3.2 Running Netfabb Simulation

3.2.1 Thermal Analysis

To run the model, from a command line run:

```
$ pan -b 03_thermal
```

The analysis progress is written on file `03_thermal.out`. To check progress run:

```
$ tail 03_thermal.out
```

or in Windows systems:

```
$ type 03_thermal.out
```

After the analysis completes, the last few lines of the output file `03_thermal.out` should be similar to the following:

```
  inc =       47 time =   14725.277     iter =   1 eps =  0.35216E+03
  inc =       47 time =   14725.277     iter =   2 eps =  0.64153E-12
 Finished writing file results/03_thermal_47.case
 Writing record:            2 , time:    14725.2769198732
 Increment end

 Mesh preview volume  =    6800.88614359419
 Activated volume     =    6800.88614359419
 Activated percentage =   100.0000000000000

 Finished writing file ./03_thermal.case

 Analysis completed

 CPU wall=   22.20703     CPU total=   72.21286
 Peak RAM used for this process = 194064 kB

 END Netfabb Simulation Solver
```

Actual CPU times will differ.

3.2.2 Quasi-Static Mechanical Analysis

Run the analysis from the command line:

```
$ pan -b 03_mechanical
```

After the analysis completes, the last few lines of the output file 03_mechanical.out should be similar to the following:

```
 *COOL time increment
----------------------
 inc =       49 time =   24445.277     iter =   1 eps =  0.52712E+06
 inc =       49 time =   24445.277     iter =   2 eps =  0.24477E-08
Finished writing file results/03_mechanical_49.case
Increment end

----------------------------------
 Substrate removal time increment
----------------------------------
 inc =       50 time =   29445.277     iter =   1 eps =  0.49409E+04
 inc =       50 time =   29445.277     iter =   2 eps =  0.52193E-09
Finished writing file results/03_mechanical_50.case
Increment end

------------------------------------------------------
 Total number of equilibrium iterations:           98

Mesh preview volume  =    6800.88614359419
Activated volume     =    6800.88614359419
Activated percentage =   100.0000000000000

Finished writing file ./03_mechanical.case

Analysis completed

************************
          1 Warning
************************

CPU wall=   47.41016     CPU total=   174.1907
Peak RAM used for this process = 728260 kB

END Netfabb Simulation Solver
```

Actual CPU times will differ.

3.3 Results

Results may be imported and viewed in Paraview or the Simulation Utility for Netfabb. Figure 3.2 shows the computed final distortion before the part is removed from the substrate.

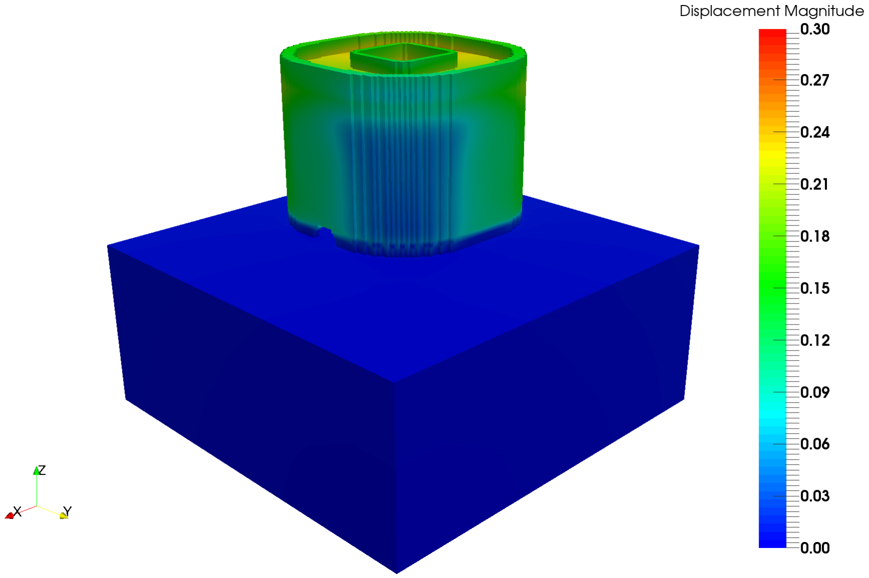


Figure 3.2: Final distortion.

There are two point cloud files produced during the mechanical simulation `03_mechanical_1.wrtu` and `03_mechanical_2.wrtu`, which are from the increments right before and after removal of the part from the build plate, respectively, for the nodes of the built component. These files have the format: [X, Y, Z, X displacement, Y displacement, Z displacement] with all the units in mm. A visualization of the X,Y,Z point cloud is shown in Figure 3.3.

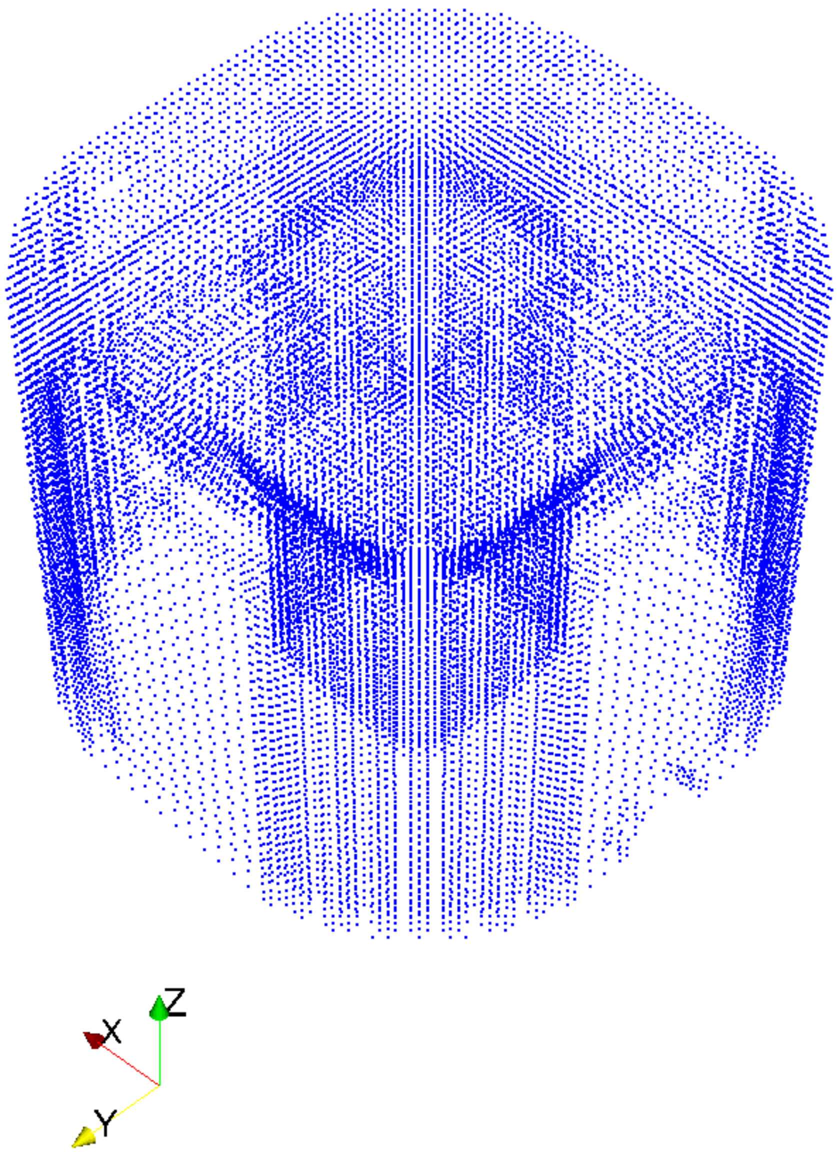


Figure 3.3: Point cloud file visualization

Example 4

Moving adaptive refinement

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

4.1 Problem Description

This is an example of moving adaptive refinement and coarsening within a layer. Only thermal analyses can be performed with this option. No element activation can currently be performed with moving adaptivity.

A moving heat source of 150 W and 600 mm/s is applied on the top surface of a 4mm x 4mm x 12.7mm substrate made of Inconel$^{\circledR}$718. A surface convection of 10.d-6 W/((mm^2)-degC) is applied on the top surface and the all other faces are insulated. The *ADPM card is used to control the acceptable temperature gradients across an element for coarsening. Increasing this number can result in artificial energy being added into the system. The mesh and laser path are automatically generated using Netfabb Simulation .

4.2 Running Netfabb Simulation

4.2.1 Thermal Analysis

From a command line run:

```
$ pan -b 04_thermal
```

The analysis progress is written on file `04_thermal.out`. To check progress run:

```
$ tail 04_thermal.out
```

or in Windows:

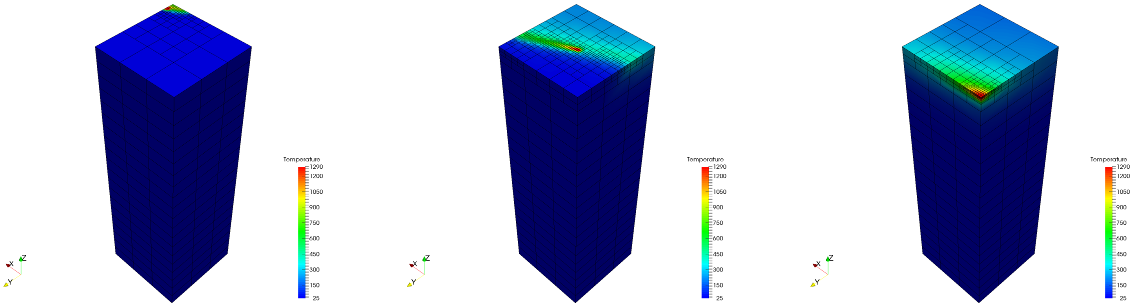
```
$ type 04_thermal.out
```

After the analysis completes, the last few lines of the output file `04_thermal.out` should be similar to the following:

```
 Sparse preprocessing complete
  inc =     1877 time =   480.00000     iter =    1 eps =  0.54104E-03
  inc =     1877 time =   480.00000     iter =   2 eps =  0.72573E-08
 Finished writing file results/04_thermal_1877.case
 Increment end
 Finished writing file ./04_thermal.case

 Analysis completed

 CPU wall=   247.3828     CPU total=   945.0073
 Peak RAM used for this process = 74876 kB

 END Netfabb Simulation Solver
```

4.3 Results

Results may be imported and viewed in Paraview or Simulation Utility for Netfabb. The results at 3 different time steps at the beginning, middle, and end of the simulation are shown in Figure 4.1



(a) First laser pass

(b) Middle laser pass

(c) Last laser pass

Figure 4.1: Temperatures at selected time increments.

Example 5

Directed Energy Deposition

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

5.1 Problem Description

This example simulates the Directed Energy Deposition (DED) build of a two bead wide, fifty layer high Ti-6Al-4V wall onto a Ti-6Al-4V substrate using the Optomec®LENS®system. The dimensions of the part are shown in Figure 5.1. For each layer, the first bead is deposited along the $+x$ direction, then the second bead is deposited in the $-x$ direction. The radius of the melt pool is 1 mm, its power is 450 W, and the translation speed is 10 mm/s. The hatch spacing between the two beads is 2 mm. The ambient temperature during the process is 30.5°C. The substrate is constrained as simply supported. The thermal and mechanical response of this process is to be calculated using Netfabb Simulation with adaptive meshing. The mesh, shown in Figure 5.2 is created automatically using the *AUTM and *SBXY cards.

5.2 Running Netfabb Simulation

5.2.1 Thermal Analysis

In the `05` directory, use a text editor to create the files named `05_thermal.in`, `05_mechanical.in` and `fin_path.lsr`. Run the analysis from the command line:

```
$ pan -b 05_thermal
```

After the analysis completes, the last few lines of the output file `05_thermal.out` should be similar to the following:

```
Finished writing file results/05_thermal_4149.case
Writing record:          83 , time:    405.711732520632
Increment end
 inc =     4150 time =   500.00000     iter =   1 eps =  0.46419E+01
 inc =     4150 time =   500.00000     iter =   2 eps =  0.14182E+01
 inc =     4150 time =   500.00000     iter =   3 eps =  0.73320E-01
 inc =     4150 time =   500.00000     iter =   4 eps =  0.19888E-03
Finished writing file results/05_thermal_4150.case
```

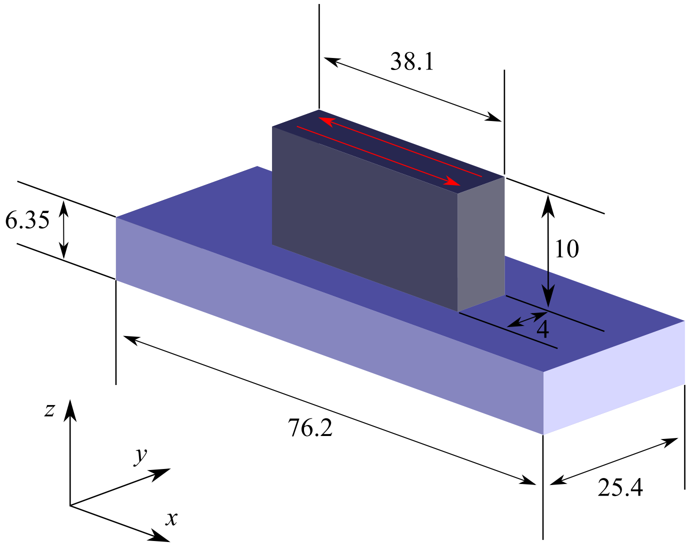


Figure 5.1: Illustration of the substrate, deposition, their dimensions in mm, the laser path (red arrows), and the coordinate system (not to scale).

```
Writing record:           84 , time:    500.000000000000
Increment end
Finished writing file ./05_thermal.case

Analysis completed

************************
          1 Warning
************************

CPU wall=   844.4414     CPU total=   3368.717
Peak RAM used for this process = 78716 kB

END Netfabb Simulation Solverabb Simulation Solver
```

5.2.2 Mechanical Analysis

Run the analysis from the command line:

```
$ pan -b 05_mechanical
```

After the analysis completes, the last few lines of the output file 05_mechanical.out should be similar to the following:

```
Increment end
 inc =     4285 time =    500.00000     iter =   1 eps =  0.70089E+05
 inc =     4285 time =    500.00000     iter =   2 eps =  0.42054E+05
 inc =     4285 time =    500.00000     iter =   3 eps =  0.45443E-09
Finished writing file results/05_mechanical_4285.case
```

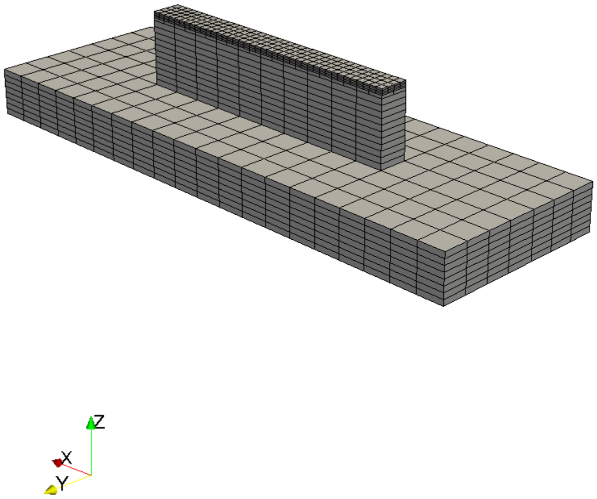
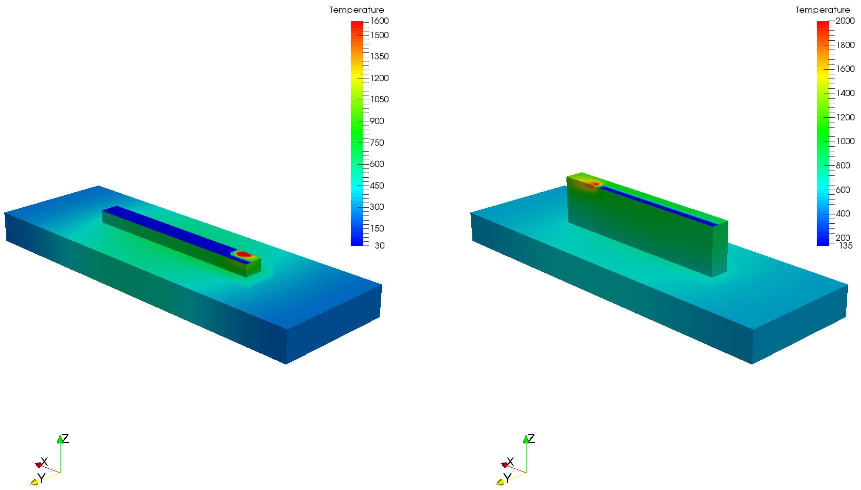


Figure 5.2: LENS part discretized by 8-node linear hexahedral elements. Dummy boundary conditions, materials, and properties must also be applied.

```
Increment end

------------------------------------------------------
 Total number of equilibrium iterations:        23916
Finished writing file ./05_mechanical.case

Analysis completed

************************
          1 Warning
************************

CPU wall=   1765.418     CPU total=   6483.370
Peak RAM used for this process = 136200 kB

END Netfabb Simulation Solver
```



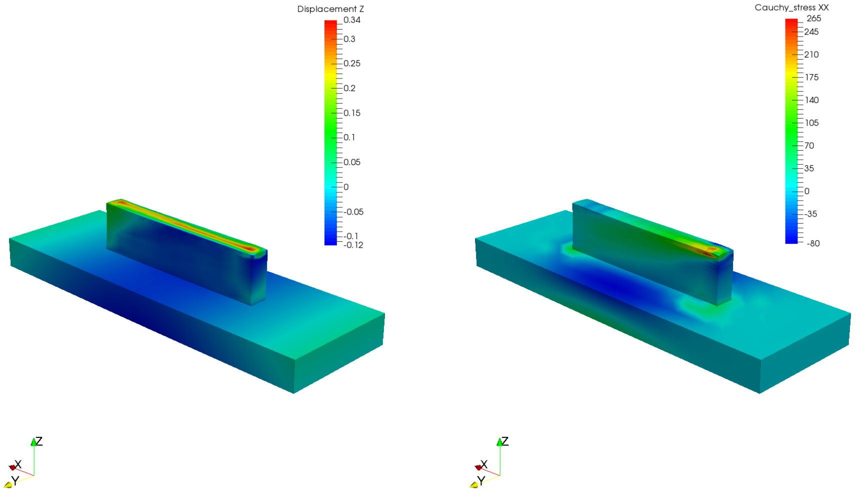
(a) Increment 1000

(b) Increment 4000

Figure 5.3: Temperature results (° C) at two sample increments.

5.3 Results

The results can be viewed in Simulation Utility for Netfabb or Paraview by importing the .case files. Thermal results during deposition are shown at two different increments in Figure 5.3. Post process distortion and a sample stress result is shown in Figure 5.4.



(a) Post Process distortion results, warped 1X

(b) Post process XX direction Cauchy stresses, warped 1X

Figure 5.4: Sample post process mechanical results

Example 6

Part Scale Modeling with Buildplate Release

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

6.1 Problem Description

A flat plate geometry of Inconel$^{\circledR}$718is built in a powder bed system and simulated. The layer height is .04mm. The part geometry is imported in the analysis through an STL file, and it is automatically meshed within Netfabb Simulation . The buildplate is modeled to be 12.7mm thick using *DDM!. The time to deposit layers is calculated using the *PBDL card. The bottom of the build plate is fixed using the *FSUB card. The *FSUB card will also simulate the release of the buildplate from the machine after the deposition process is complete, but before the part is removed from the buildplate. The resulting mesh is illustrated in Figures 6.1.

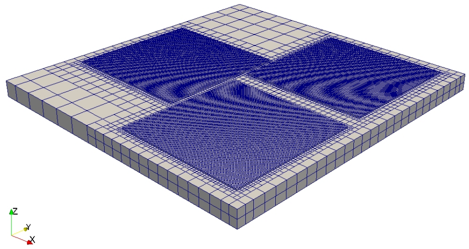


Figure 6.1: Auto-generated voxel mesh.

A time incremental thermal analysis is performed first to compute the temperature history of the part. Layers are activated in groups using *PBPA, and additional time increments are used to model heat conduction into the part. The thermal analysis includes only the part and substrate. Heat loss into the powder is modeled as convection with a value of 25.d-6 W/((mm^2)°C) using *CONV.

A time incremental mechanical analysis is performed after the thermal analysis is completed. Similarly to the thermal analysis, layers are activated in groups and the computed temperature distribution from the mechanical analysis is used to compute deformation due to the thermal expansion.

6.2 Running Netfabb Simulation

6.2.1 Thermal Analysis

To run the model in batch mode, on a windows machine run from the command line:

```
$ pan -q 06_win
```

If the system is linux, from the terminal run:

```
$ pan -q 06_linux
```

These files are identical in essence, but linux and windows do not have interchangeable text file formats.

This will run the input files in the .que file in series. The input files must be in the same folder as the .que file. This allows users to easily simulate large batches of jobs from the command line.

The thermal analysis progress is written on file `06_thermal.out`. To check progress run:

```
$ tail 06_thermal.out
```

or in Windows systems:

```
$ type 06_thermal.out
```

After the analysis completes, the last few lines of the output file `06_thermal.out` should be similar to the following:

```
Increment end
 inc =       17 time =   57972.801     iter =   1 eps =  0.83376E+00
 inc =       17 time =   57972.801     iter =   2 eps =  0.48291E-13
Finished writing file results/06_thermal_17.case
Writing record:            2 , time:    57972.8014167935
Increment end

Mesh preview volume  =    190987.524817100
Activated volume     =    190987.524817100
Activated percentage =    100.000000000000

Finished writing file ./06_thermal.case

Analysis completed

************************
          1 Warning
************************
```

```
CPU wall=   49.00391     CPU total=   124.1144
Peak RAM used for this process = 610476 kB

END Netfabb Simulation Solver
```

6.2.2 Quasi-Static Mechanical Analysis

After the thermal analysis completes, the que file with automatically run the mechanical analysis immediately afterwards.

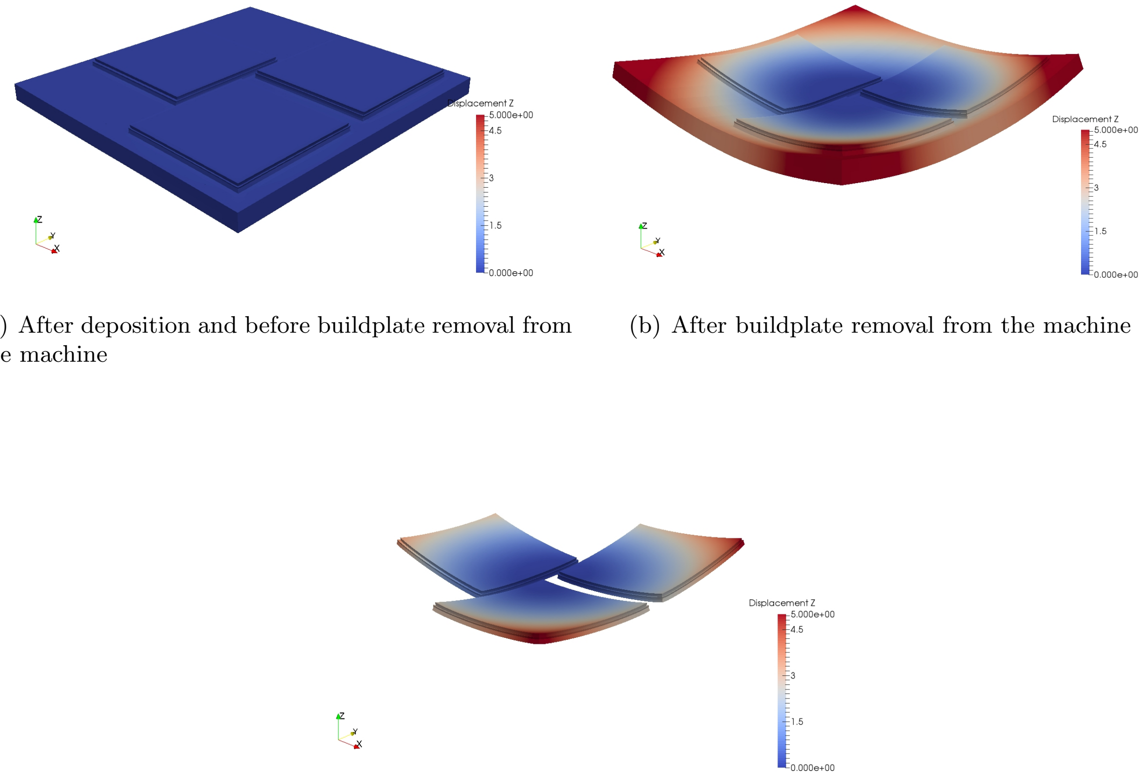
After the mechanical analysis completes, the last few lines of the output file `06_mechanical.out` should be similar to the following:

```
----------------------------------
 Substrate removal time increment
----------------------------------
 inc =       20 time =   207672.80     iter =   1 eps =  0.21556E+06
 inc =       20 time =   207672.80     iter =   2 eps =  0.14109E-07
Finished writing file results/06_mechanical_20.case
Increment end

------------------------------------------------------
 Total number of equilibrium iterations:           33

Mesh preview volume  =    190987.524817100
Activated volume     =    190987.524817100
Activated percentage =    100.000000000000

Finished writing file ./06_mechanical.case

Analysis completed

************************
          2 Warnings
************************

CPU wall=   112.1367     CPU total=   382.1245
Peak RAM used for this process = 3125884 kB

END Netfabb Simulation Solver
```

6.3 Results

Results may be imported and viewed in Simulation Utility for Netfabb or Paraview.

Figures 6.2 shows the computed final distortion before the buildplate is removed from the machine, after the buildplate is removed from the machine, and after the part is removed from the buildplate.



(a) After deposition and before buildplate removal from the machine

(b) After buildplate removal from the machine

(c) After part removal from the buildplate

Figure 6.2: Distortion [mm] (5x magnification).

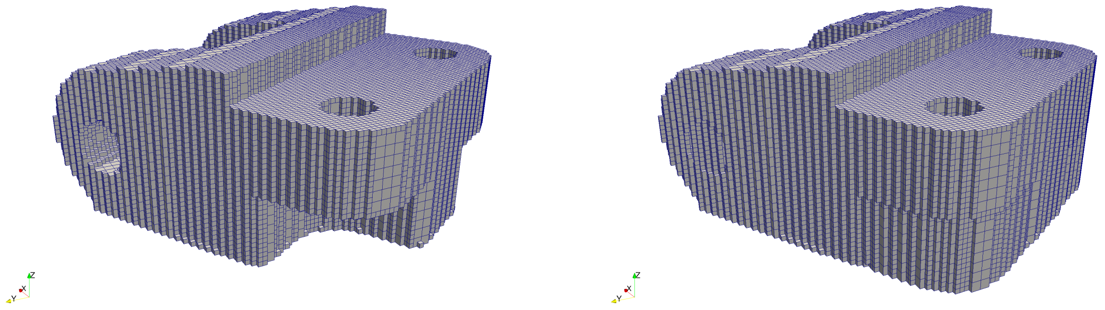
Example 7

Part Scale Modeling with CLI Support Structures

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

7.1 Problem Description

An Inconel® 718 bracket geometry with support structures is built in a powder bed system. The layer height is 0.04mm. The part geometry is imported in the analysis through an STL file, the support structure is imported from an CLI file, and both are automatically meshed within Netfabb Simulation . The buildplate is modeled to be 25.4 mm thick using *DDM!. The time to deposit layers is calculated using the *PBDL card. The bottom of the build plate is fixed using the *FSUB card. The *FSUB card will also simulate the release of the buildplate from the machine after the deposition process is complete, but before the part is removed from the buildplate. The mesh, shown with and without the support elements, is shown in Figures 7.1.



(a) Bracket without supports

(b) Bracket with supports

Figure 7.1: Bracket with and without supports.

A time incremental thermal analysis is performed first to compute the temperature history of the part. Layers are activated in groups using *PBPA, and additional time increments are used to model heat conduction into the part. The thermal analysis includes only the part and substrate.

Heat loss into the powder is modeled as convection with a value of 25.d-6 W/((mm^2)°C) using *CONV.

A time incremental mechanical analysis is performed after the thermal analysis is completed. Similarly to the thermal analysis, layers are activated in groups and the computed temperature distribution from the mechanical analysis is used to compute deformation due to the thermal expansion. These simulations have three additional post-process simulation increments, first Netfabb Simulation simulates the release of the buildplate from the machine, then the removal of the build from the buildplate, and finally the removal of the support structure material from the final build.

Two variations of the mechanical analysis is performed, a basic analysis (07_mechanical1.in), and an advanced analysis which simulates failure at the support-build interface (07_mechanical2.in).

7.2 Running Netfabb Simulation

7.2.1 Thermal Analysis

To run the model, from a command line run:

```
$ pan -b 07_thermal
```

The analysis progress is written on file `07_thermal.out`. To check progress run:

```
$ tail 07_thermal.out
```

After the analysis completes, the last few lines of the output file `07_thermal.out` should be similar to the following:

```
Increment end
 inc =       29 time =   110026.96     iter =   1 eps =  0.12597E+04
 inc =       29 time =   110026.96     iter =   2 eps =  0.10635E-11
Finished writing file results/07_thermal_29.case
Writing record:            2 , time:    110026.958120637
Increment end

Mesh preview volume  =    110852.789200311
Activated volume     =    110852.789200311
Activated percentage =    100.000000000000

Finished writing file ./07_thermal.case

Analysis completed

CPU wall=   65.23438     CPU total=   168.0443
Peak RAM used for this process = 325668 kB

END Netfabb Simulation Solver
```

Actual CPU times will differ.

7.2.2 Quasi-Static Mechanical Analysis

Run the first mechanical analysis from the command line:

```
$ pan -b 07_mechanical1
```

After the analysis completes, the last few lines of the output file `07_mechanical1.out` should be similar to the following:

```
------------------------------------------
 Support structure removal time increment
------------------------------------------
 inc =       33 time =   309776.96     iter =   1 eps =  0.17251E+05
 inc =       33 time =   309776.96     iter =   2 eps =  0.38989E-08
Finished writing file results/07_mechanical1_33.case
Increment end

------------------------------------------------------
 Total number of equilibrium iterations:           56

Mesh preview volume  =    110852.789200311
Activated volume     =    110852.789200311
Activated percentage =    100.000000000000

Finished writing file ./07_mechanical1.case

Analysis completed

************************
          1 Warning
************************

CPU wall=   85.37500     CPU total=   292.3147
Peak RAM used for this process = 1234932 kB

END Netfabb Simulation Solver
```

Actual CPU times will differ.

Run the second mechanical analysis from the command line:

```
$ pan -b 07_mechanical2
```

After the analysis completes, the last few lines of the output file `07_mechanical2.out` should be similar to the following:

```
*** WARNING:           24
Support structure failure
-------------------------------
Increment end

------------------------------------------------------
```

```
  Total number of equilibrium iterations:          65

 Mesh preview volume  =    110852.789200311
 Activated volume     =    110852.789200311
 Activated percentage =    100.000000000000

 Finished writing file ./07_mechanical2.case

 Analysis completed

 ************************
          24 Warnings
 ************************

 CPU wall=   100.3477     CPU total=   343.8574
 Peak RAM used for this process = 1231424 kB

 END Netfabb Simulation Solver
```

Each of the warnings note a support structure element failure.

7.3 Results

There are two additional log files created during simulation, file_name_recoater.txt and file_name_warning.txt.

For case 07_mechanical1 there are no warnings, so the 07_mechanical1_warnings.txt file is empty. However the recoater clearance is written. Below are the results seen in the 07_mechanical1_recoater.txt:

```
time (s), layer group, recoater clearance (%), top z deformed coord (mm), recoater coord (mm)
  1.172554E+04             1        77.854  4.008859E+00  4.040000E+00
  2.389416E+04             2        58.331  8.016668E+00  8.040000E+00
  3.607386E+04             3        46.876  1.202125E+01  1.204000E+01
  4.815440E+04             4        70.065  1.601197E+01  1.604000E+01
  6.195965E+04             5        64.048  2.001438E+01  2.004000E+01
  7.656102E+04             6        59.386  2.401625E+01  2.404000E+01
  9.112844E+04             7        59.090  2.801636E+01  2.804000E+01
  1.035748E+05             8        78.648  3.200854E+01  3.204000E+01
  1.092742E+05             9        79.082  3.600837E+01  3.604000E+01
```

Now are the results from the 07_mechanical1_recoater.txt file:

```
time (s), layer group, recoater clearance (%), top z deformed coord (mm), recoater coord (mm)
  1.172554E+04             1        77.854  4.008859E+00  4.040000E+00
  2.389416E+04             2        58.331  8.016668E+00  8.040000E+00
  3.607386E+04             3        46.876  1.202125E+01  1.204000E+01
  4.815440E+04             4        70.066  1.601197E+01  1.604000E+01
```

```
 6.195965E+04             5        64.053  2.001438E+01  2.004000E+01
 7.656102E+04             6        58.857  2.401646E+01  2.404000E+01
 9.112844E+04             7        58.826  2.801647E+01  2.804000E+01
 1.035748E+05             8        78.461  3.200862E+01  3.204000E+01
 1.092742E+05             9        79.083  3.600837E+01  3.604000E+01
```

Neither of these simulations show any indication that the recoater blade will experience interference.

Below are the contents of the 07_mechanical2_warning.txt file.

```
-------------------------------

*** WARNING:            1
Support structure failure
-------------------------------

*** WARNING:            2
Time is too close to previous time for
postprocessing.  Postprocessing will show
artificially increased times from this
increment onwards.

The .out file will still show correct times

-------------------------------

*** WARNING:            3
Support structure failure
-------------------------------
-------------------------------

*** WARNING:            4
Support structure failure
-------------------------------
-------------------------------

*** WARNING:            5
Support structure failure
-------------------------------
-------------------------------

*** WARNING:            6
Support structure failure
-------------------------------
-------------------------------

*** WARNING:            7
Support structure failure
```

```
-------------------------------
-------------------------------

*** WARNING:            8
Support structure failure
-------------------------------
-------------------------------

*** WARNING:            9
Support structure failure
-------------------------------
-------------------------------

*** WARNING:           10
Support structure failure
-------------------------------
-------------------------------

*** WARNING:           11
Support structure failure
-------------------------------
-------------------------------

*** WARNING:           12
Support structure failure
-------------------------------
-------------------------------

*** WARNING:           13
Support structure failure
-------------------------------
-------------------------------

*** WARNING:           14
 Recoater clearance =     1.%
 Top z deformed coord:   20.0397    Recoater coord:   20.0400
-------------------------------
-------------------------------

*** WARNING:           15
Support structure failure
-------------------------------
-------------------------------

*** WARNING:           16
Support structure failure
-------------------------------
```

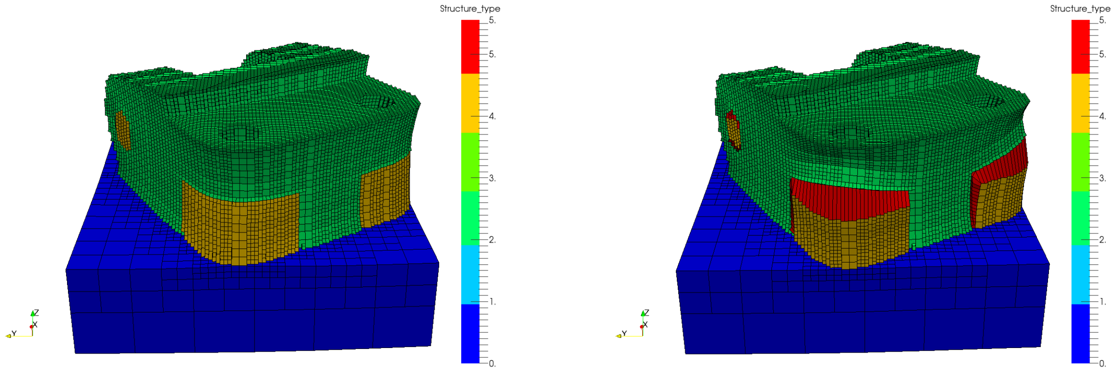
```
-------------------------------

*** WARNING:           17
Support structure failure
-------------------------------
-------------------------------

*** WARNING:           18
 Recoater clearance =   -23.%
 Top z deformed coord:   24.0490    Recoater coord:   24.0400
-------------------------------
-------------------------------

*** WARNING:           19
Support structure failure
-------------------------------
-------------------------------

*** WARNING:           20
Support structure failure
-------------------------------
-------------------------------

*** WARNING:           21
Support structure failure
-------------------------------
-------------------------------

*** WARNING:           22
 Recoater clearance =    11.%
 Top z deformed coord:   28.0355    Recoater coord:   28.0400
-------------------------------
-------------------------------

*** WARNING:           23
Support structure failure
-------------------------------
-------------------------------

*** WARNING:           24
Support structure failure
-------------------------------
```

Observing this file will indicate significant support structure failure. This will be apparent when we look into the results in a visualization program.

There is another feature that enhances the user's ability to investigate support structure failure, an output called Structure Type, which may be viewed in post processing software. This result

is assigns an integer value to each element, indicating what kind of structure the element is. The values, ranging from 0-5, are as follows:
0 - Build plate
1 - Powder
2 - Component
3 - Homogenized component
4 - Support structure
5 - Failed support structure

Figures 7.2 shows the results for cases 07_mechanical1 and 07_mechanical2 at the 3rd to the last increment, while the part and supports are still attached to the build plate. The results have been warped by displacements which are magnified 5x to better show the effect of failed elements.



(a) Mechanical case 1 structure type

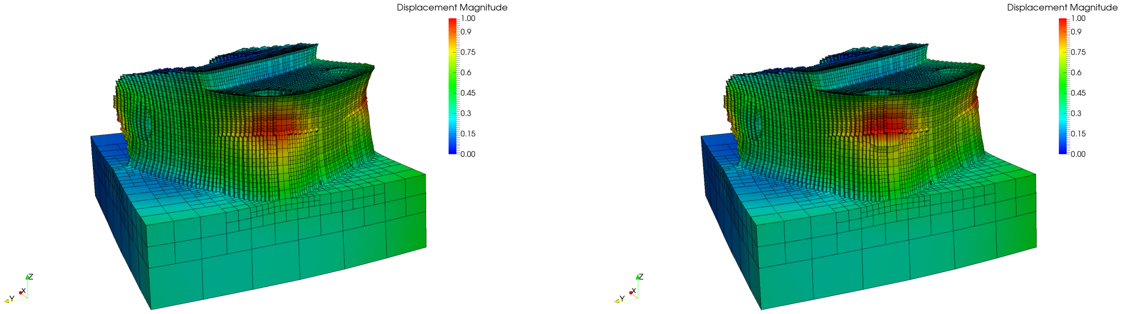
(b) Mechanical case 2 structure type

Figure 7.2: Structure type results

For case 07_mechanical1, seen in Figure 7.2(a), elements only have values 0-4, and all support structures are coded as type 4, unfailed supports. This is as expected as this analysis does not have the support structure card *UTSR enabled. For case 07_mechanical2, shown in Figure 7.2(b), there are numerous failed supports, as indicated in both the simulation log file and the 07_mechanical2_warning.txt file. Note how the failed elements are stretched. This occurs because the failed elements no longer have any strength to resist deformation.

Now examine and compare the displacement results from the two mechanical cases. Figures 7.3 depicts the distortion of the two cases while the supports and component are attached to the build plate, again warped by a 5x magnification factor.

Figures 7.3 shows the computed final distortion from the support structure failure analysis using *UTSR (07_mechanical2.in) after the part is removed from the buildplate, and after the support material has been removed. Note the increase of distortion and elongation of failed elements. Support structure failure and the resulting displacement can be mitigated by increasing the density of the support structure, changing the orientation of parts to avoid or reduce overhangs, or changes to the build geometry itself.



(a) Mechanical case 1

(b) Mechanical case 2

Figure 7.3: Distortion [mm] (10x magnification) of the two mechanical cases.

Example 8

Powder Bed Moving Source Modeling with Custom Toolpaths

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

8.1 Problem Description

A set of 4 Inconel$^{®}$ 718 analyses are included in this example which describe the modeling of powder bed tracks using the moving source method. The custom build geometry is defined by the laser path file **`thft_line1.lsr`** which is called by the *LSRF card. The example laser path includes 3 deposition tracks 5 mm long and 0.04 mm high. The substrate thickness is defined by the second value in the DDM! card and the substrate X and Y is extended 10 mm in all 4 planar directions from the bounding box defined by the build geometry.

The 4 thermal analyses which comprise this example which are:

- t.in - Simulates substrate heating using a static autogenerated mesh
- tadpm.in - Simulates substrate heat using a moving-adaptive autogenerated mesh
- tdirect.in - Simulates powder sintering using a static autogenerated mesh
- tddmp.in - Simulates powder sintering using a static autogenerated mesh and active powder elements outside of the 3 deposition tracks

8.2 Running Netfabb Simulation

8.2.1 Thermal Analysis

To run the t.in model, from a command line run:

```
$ pan -b t
```

The analysis progress is written on file `t.out`. To check progress run:

```
$ tail t.out
```

or in Windows run:

```
$ type t.out
```

After the analysis completes, the last few lines of the output file `t.out` should be similar to the following:

```
 Increment end
 Increment end
  inc =      247 time =   60.000000     iter =   1 eps =  0.46496E-03
 Finished writing file results/t_247.case
 Increment end
 Finished writing file ./t.case

 Analysis completed

 CPU wall=   106.3984     CPU total=   2064.754
 END Netfabb Simulation Solver
```

Actual CPU times will differ. To run the tadpm.in model, from a command line run:

```
$ pan -b tadpm
```

After the analysis completes, the last few lines of the output file `tadpm.out` should be similar to the following:

```
 Starting auxspar
 Number of no zeros nsymmetric =      136319
 Sparse preprocessing complete
  inc =      244 time =   60.000000     iter =   1 eps =  0.37673E-03
 Finished writing file results/tadpm_244.case
 Increment end
 Finished writing file ./tadpm.case

 Analysis completed

 CPU wall=   63.60547     CPU total=   1269.486
 END Netfabb Simulation Solver
```

Actual CPU times will differ. To run the tdirect.in model, from a command line run:

```
$ pan -b tdirect
```

After the analysis completes, the last few lines of the output file `tdirect.out` should be similar to the following:

```
 Increment end
  inc =      247 time =   60.000000     iter =   1 eps =  0.95290E-03
 Finished writing file results/tdirect_247.case
 Increment end
 Finished writing file ./tdirect.case
```

```
Analysis completed

CPU wall=   79.41016     CPU total=   1585.364
END Netfabb Simulation Solver
```

Actual CPU times will differ. To run the tddmp.in model, from a command line run:

```
    $ pan -b tddmp
```

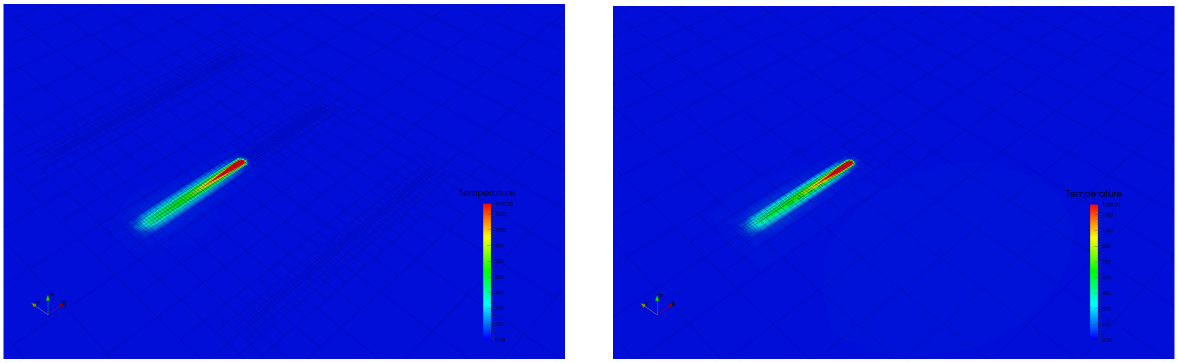
After the analysis completes, the last few lines of the output file `tddmp.out` should be similar to the following:

```
Increment end
 inc =       244 time =   60.000000     iter =   1 eps =  0.37967E-03
Finished writing file results/tddmp_244.case
Increment end
Finished writing file ./tddmp.case

Analysis completed

CPU wall=   136.4219     CPU total=   2725.084
END Netfabb Simulation Solver
```

Actual CPU times will differ.

8.3 Results

Results may be imported and viewed in the Simulation Utility for Netfabb or Paraview. Figures 8.1 shows the results of each of the 2 substrate thermal simulations and Figures 8.2 shows the results of each of the 2 powder sintering thermal simulations, each during the simulation of the second laser scan.

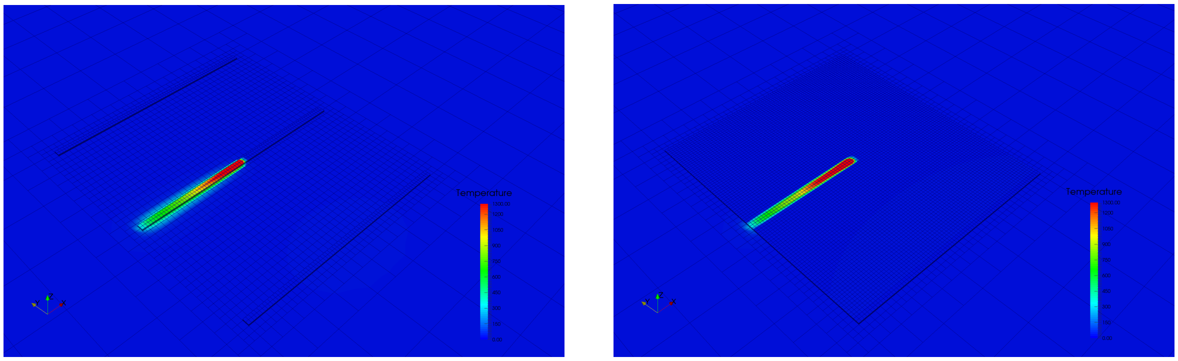


(a) t.in

(b) tadpm.in

Figure 8.1: Thermal simulation of substrate heating

The substrate heating simulations both model the thermal behavior during moving heat source based pre-heating. Using the fixed autogenerated mesh, shown in Figure 8.1(a) produces a fine



(a) tdirect.in

(b) tddmp.in

Figure 8.2: Thermal simulation of powder sintering

mesh in the regions of heating. The adaptive mesh shows only fine mesh around the heat source at the individual time increment in Figure 8.1(b).

Powder sintering without modeling the powder is shown in Figure 8.2(a) while the simulation including the powder is shown in Figure 8.2(b).

Example 9

Powder Bed Part Level Plasticity

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

9.1 Problem Description

A generic geometry of Inconel$^{®}$625 is built in a powder bed system and simulated. The layer height is 0.04mm. The part geometry is imported in the analysis through an STL file, and it is automatically meshed by the Netfabb Simulation solver. The substrate is assumed to be 38.1mm thick. The resulting mesh is illustrated in Figures 9.1.

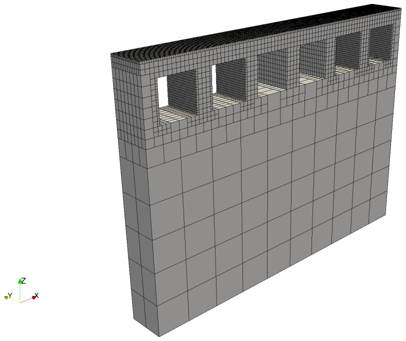


Figure 9.1: Autogenerated mesh.

A time incremental thermal analysis is performed first to compute the temperature history of the part. Layers are activated in groups, and additional time increments are used to model heat conduction into the part. The thermal analysis includes only the part and substrate. Heat loss into the powder is modeled as convection with a value of 25.d-6 W/((mm^2)°C) using the *CONV option.

Two time incremental mechanical analyses are performed after the thermal analysis is completed, one with qualitative stresses, one with quantitative stresses. Similarly to the thermal analysis, layers are activated in groups using *PBPA and the computed temperature distribution from the mechanical analysis is used to compute deformation due to the thermal expansion. The source PRM file is for Inconel 625, using generic processing parameters.

9.2 Running Netfabb Simulation

9.2.1 Thermal Analysis

To run the model, from a command line run:

```
$ pan -b 09_thermal
```

The analysis progress is written on file 09_thermal.out. To check progress run:

```
$ tail 09_thermal.out
```

After the analysis completes, the last few lines of the output file 09_thermal.out should be similar to the following:

```
 Sparse preprocessing complete
 results file name =results/09_thermal_00013.bin
  inc =       25 time =   1894.6941     iter =   1 eps =  0.26475E+03
  inc =       25 time =   1894.6941     iter =   2 eps =  0.73228E+02
  inc =       25 time =   1894.6941     iter =   3 eps =  0.20452E-12
 Writing record:            1 , time:    1894.69411764706
 Increment end
  inc =       26 time =   1982.8118     iter =   1 eps =  0.14964E+03
  inc =       26 time =   1982.8118     iter =   2 eps =  0.61717E-12
 Finished writing file results/09_thermal_26.case
 Writing record:            2 , time:    1982.81176470588
 Increment end

 Mesh preview volume  =    791.560000000000
 Activated volume     =    791.560000000000
 Activated percentage =   100.0000000000000

 Finished writing file ./09_thermal.case

 Analysis completed

 CPU wall=   11.49219     CPU total=   33.19701
 Peak RAM used for this process = 153668 kB

 END Netfabb Simulation Solver
```

Actual CPU times will differ.

9.2.2 Quasi-Static Mechanical Analysis

Run the first analysis from the command line:

```
$ pan -b 09_mechanical
```

The analysis progress is written on file 09_mechanical.out. To check progress run:

```
$ tail 09_mechanical.out
```

After the analysis completes, the last few lines of the output file 09_mechanical.out should be similar to the following:

```
 ----------------------------------
 Substrate removal time increment
----------------------------------
 inc =       29 time =   151912.81     iter =   1 eps =  0.38866E+04
 inc =       29 time =   151912.81     iter =   2 eps =  0.23759E-08
Finished writing file results/09_mechanical_29.case
Increment end

------------------------------------------------------
 Total number of equilibrium iterations:           57

Mesh preview volume  =    791.560000000000
Activated volume     =    791.560000000000
Activated percentage =   100.0000000000000

Finished writing file ./09_mechanical.case

Analysis completed

CPU wall=   20.84375     CPU total=   69.26444
Peak RAM used for this process = 584724 kB

END Netfabb Simulation Solver
```

Actual CPU times may differ.

Run the second analysis from the command line:

```
$ pan -b 09_mechanical_ppla
```

The analysis progress is written on file 09_mechanical_ppla.out. To check progress run:

```
$ tail 09_mechanical_ppla.out
```

After the analysis completes, the last few lines of the output file 09_mechanical_ppla.out should be similar to the following:

```
------------------------------------
 Plasticity iteration #           1
------------------------------------
 inc =       29 time =   151912.81     iter =   1 eps =  0.34141E+04
------------------------------------
 Plasticity iteration #           2
------------------------------------
 inc =       29 time =   151912.81     iter =   1 eps =  0.34141E+04
 inc =       29 time =   151912.81     iter =   2 eps =  0.12592E+04
```

```
  inc =       29 time =   151912.81     iter =   3 eps =  0.40049E+03
  inc =       29 time =   151912.81     iter =   4 eps =  0.99917E+01
  inc =       29 time =   151912.81     iter =   5 eps =  0.59297E-01
  inc =       29 time =   151912.81     iter =   6 eps =  0.28033E-05
 Finished writing file results/09_mechanical_ppla_29.case
 Increment end

 ----------------------------------
  Substrate removal time increment
 ----------------------------------
  inc =       30 time =   201912.81     iter =   1 eps =  0.29687E+04
  inc =       30 time =   201912.81     iter =   2 eps =  0.17267E-08
 Finished writing file results/09_mechanical_ppla_30.case
 Increment end

 ------------------------------------------------------
  Total number of equilibrium iterations:          64

 Mesh preview volume  =    791.560000000000
 Activated volume     =    791.560000000000
 Activated percentage =   100.0000000000000

 Finished writing file ./09_mechanical_ppla.case

 Analysis completed

 CPU wall=   25.89062     CPU total=   89.37298
 Peak RAM used for this process = 735160 kB

 END Netfabb Simulation Solver
```

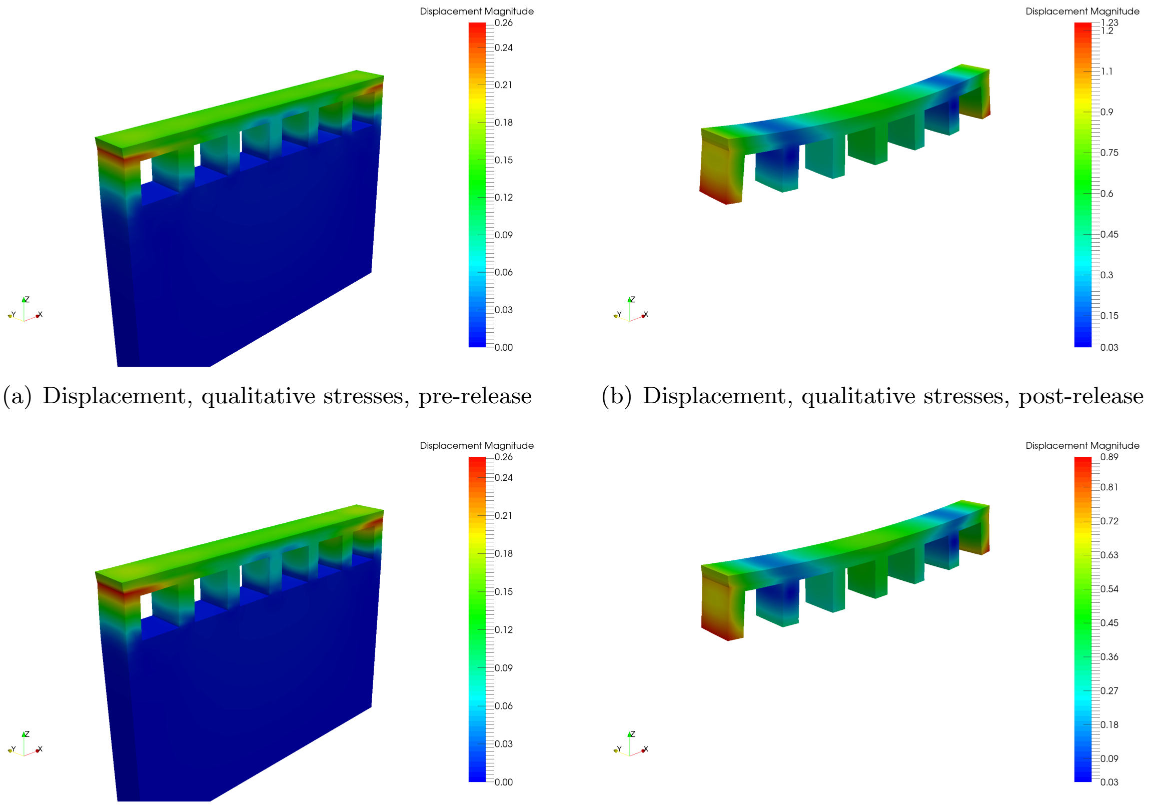
Actual CPU times may differ. Note the plasticity steps at the end of the simulation, after the *COOL time increment and before the Substrate removal time increment.

9.3 Results

Results may be imported and viewed in Paraview or Simulation Utility for Netfabb.

Figures 9.2 shows the computed distortions before and after substrate release for both mechanical analyses.

Observe that pre-release distortions are roughly equivalent for the two cases. However, post release, the quantitative stress case exhibits displacements which are 28% less than without using plasticity. This shows the necessity of accounting for this behavior when looking at post-EDM builds which do not undergo heat treating.



(a) Displacement, qualitative stresses, pre-release

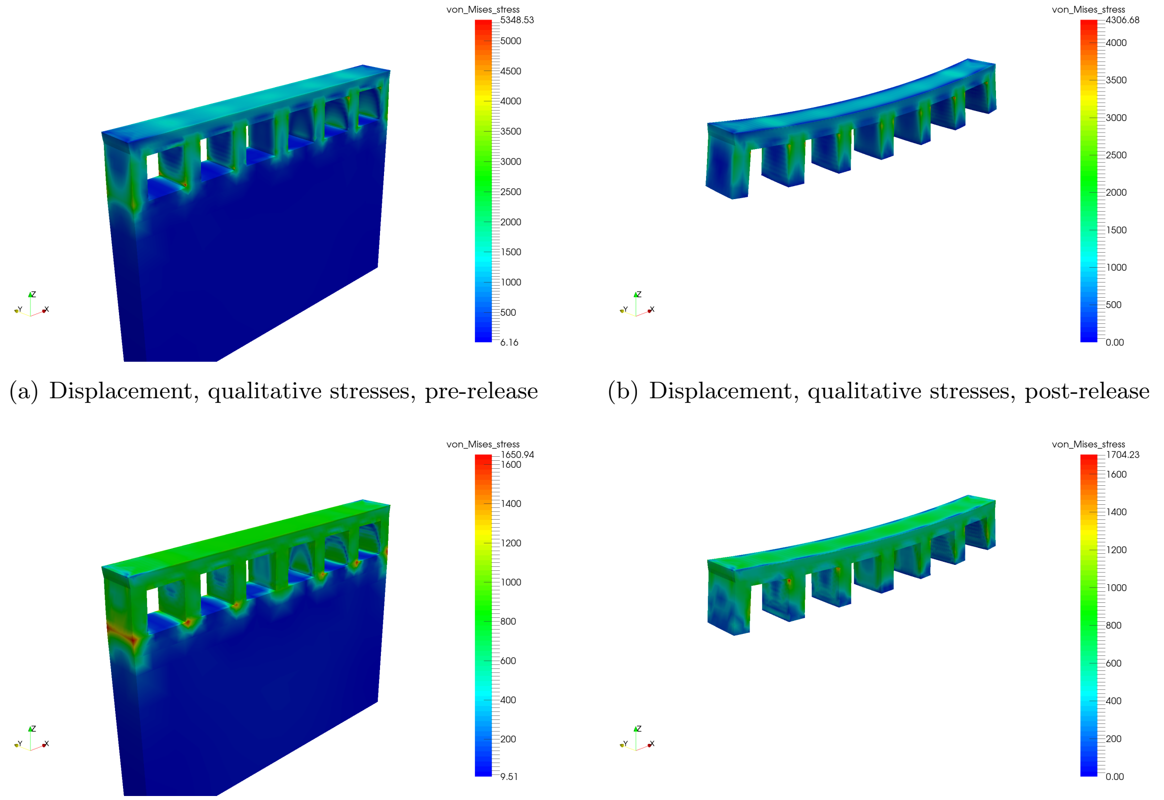
(b) Displacement, qualitative stresses, post-release

(c) Displacement, quantitative stresses, pre-release

(d) Displacement, quantitative stresses, post-release

Figure 9.2: Displacement results

Figures 9.3 shows the computed distortions before and after substrate release for both mechanical analyses.



(a) Displacement, qualitative stresses, pre-release

(b) Displacement, qualitative stresses, post-release

(c) Displacement, quantitative stresses, pre-release

(d) Displacement, quantitative stresses, post-release

Figure 9.3: Von Mises stress results

Observe that the qualitative stresses are unrealistically high, but still indicate the same regions of peak stress in the quantitative stress case.

Example 10

Lack of Fusion *LFUS and *TPRE example

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

10.1 Problem Description

This is an example of using the state variable cards *LFUS and *TPRE to inspect the thermal history of 2 moving source simulations. The first simulation is the analysis of a single powder bed layer using the adaptive refinement methods described in Example 4. The second simulation is the analysis of a multilayer powder bed analysis using a moving heat source using both layerwise and moving adaptivity mesh coarsening techniques.

For the single layer moving adaptivity simulation, a moving heat source of 50W moving at 1000 mm/s is applied on the top surface of a 1.0mm × 1.0mm × 12.7mm substrate made of Ti-6Al-4V. A surface convection of 10.d-6 W/((mm^2)°C) is applied on the top surface and the all other faces are insulated. The *ADPM card is used to control the acceptable temperature gradients across an element for coarsening, using the default settings. The mesh and laser path are automatically generated by the Netfabb Simulation solver. The melting of 3 powder layers are simulated.

For the multilayer adaptivity simulation a 3 layer simulation is completed on the top surface of a 0.5mm × 0.5mm × 12.7 mm Ti-6Al-4V substrate, with a 25.d-6 W/((mm^2)°C) top surface convection, also using 50 W heat source and a scan speed of 1000 mm/s, with a 120 s interlayer dwell. Layerwise adaptivity is controlled on the auto-generated mesh using the *ADAP and *ADP1 cards. The *ADPM card is used to enable moving adaptivity.

For both simulations the *LFUS card is used with a value of 1600°C, to investigate lack of fusion. The *TPRE card is used to inspect the temperatures immediately prior to application of the heat source with the values of 690 and 1600°C, which are the stress relaxation and melting temperatures respectively.

10.2 Running Netfabb Simulation

10.2.1 Moving Adaptivity Thermal Analysis

From a command line run:

```
$ pan -b t10_moving
```

The analysis progress is written on file `moving_adapt.out`. To check progress run:

```
$ tail t10_moving.out
```

After the analysis completes, the last few lines of the output file `moving_adapt.out` should be similar to the following:

```
 Sparse preprocessing complete
  inc =      650 time =   480.00000     iter =   1 eps =  0.67771E-04
 Finished writing file results/t10_moving_650.case
 Finished writing file results/t10_moving_650_c.case
 Increment end
 Finished writing file ./t10_moving.case
 Finished writing file ./t10_moving_c.case

 Analysis completed

 CPU wall=   390.3320     CPU total=   1545.673
 Peak RAM used for this process = 144144 kB

 END Netfabb Simulation Solver
```

Actual CPU times will differ.

10.2.2 Multilayer Thermal Analysis

From a command line run:

```
$ pan -b t10_multilayer
```

The analysis progress is written on file `multilayer_adapt.out`. To check progress run:

```
$ tail t10_multilayer.out
```

After the analysis completes, the last few lines of the output file `multilayer_adapt.out` should be similar to the following:

```
 Sparse preprocessing complete
  inc =      231 time =   800.00000     iter =   1 eps =  0.60697E-02
 Finished writing file results/t10_multilayer_231.case
 Finished writing file results/t10_multilayer_231_c.case
 Increment end
 Finished writing file ./t10_multilayer.case
 Finished writing file ./t10_multilayer_c.case

 Analysis completed

 CPU wall=   51.80469     CPU total=   200.9605
 Peak RAM used for this process = 105040 kB
```

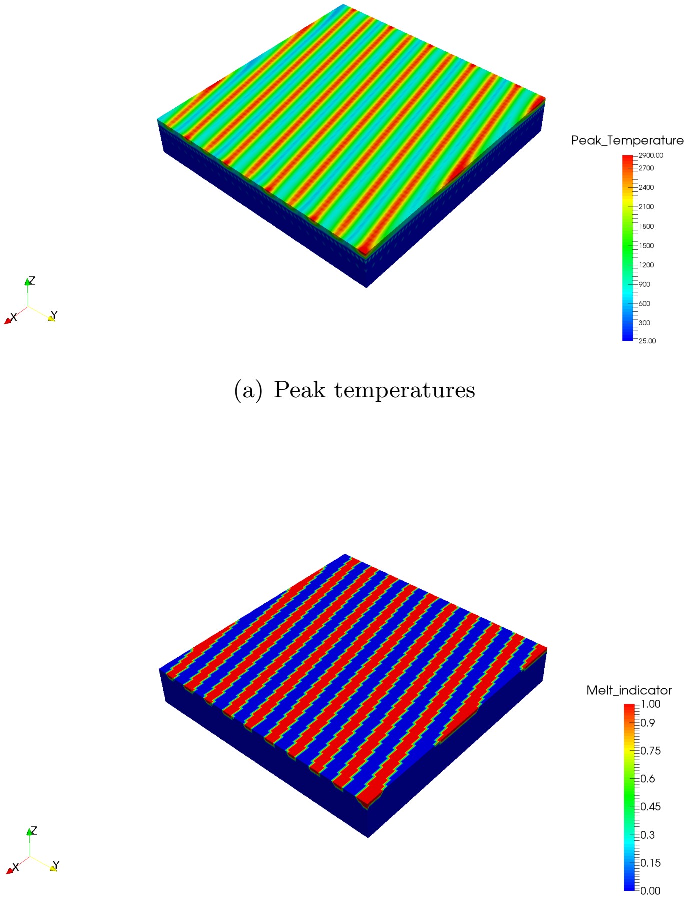
Actual CPU times will differ.

10.3 Results

Results may be imported and viewed in Paraview.

10.3.1 Moving Adaptivity Thermal Analysis

Figure 10.1 shows the results of the two state variables added to the Paraview results when using *LFUS in the moving adaptivity simulation.



(a) Peak temperatures

(b) Melt indicator

Figure 10.1: *LFUS results moving adaptive mesh

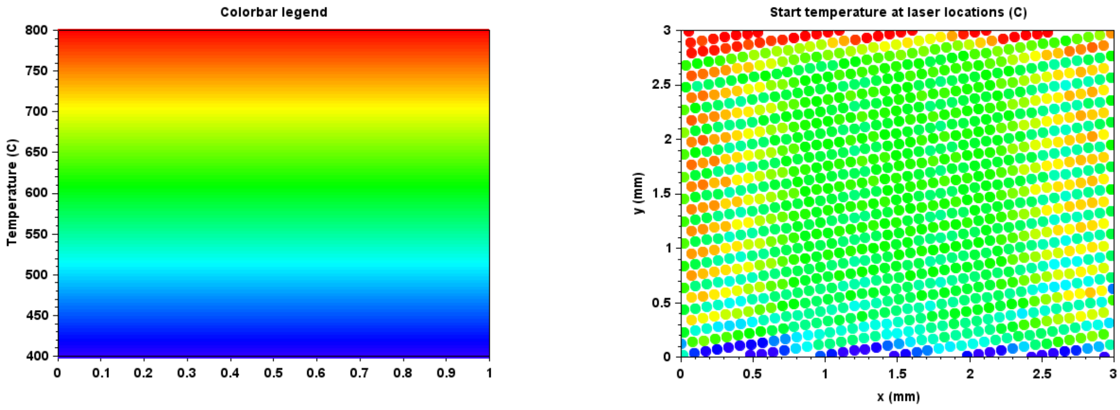
Using *TPRE in this simulation also produces a file `t10moving_tpre.txt`, which gives the increment time, heat source location, preheat temperature at that heat source location, and binary flags (1 for those that exceed the test temperatures, 0 for those which fall below) for the temperatures of interest.

10.3.2 *TPRE results file `t10moving_tpre.txt`

These are the results for the first 40 time increments

```
# time, laser_x, laser_y, laser_z, Temp_start, StateVar1 for Temp_crit =   6.900000E+02,   1.600000E+03,
  2.110243E-04,   9.812045E-01,   6.025919E-02,   0.000000E+00,   1.183370E+03,            1,           0,
  2.110243E-04,   9.812045E-01,   6.025919E-02,   0.000000E+00,   1.183370E+03,            1,           0,
  2.310243E-04,   9.616060E-01,   5.627184E-02,   0.000000E+00,   1.246882E+03,            1,           0,
  2.310243E-04,   9.616060E-01,   5.627184E-02,   0.000000E+00,   1.246882E+03,            1,           0,
  2.510243E-04,   9.420075E-01,   5.228448E-02,   0.000000E+00,   1.264094E+03,            1,           0,
  2.510243E-04,   9.420075E-01,   5.228448E-02,   0.000000E+00,   1.264094E+03,            1,           0,
  2.710243E-04,   9.224090E-01,   4.829712E-02,   0.000000E+00,   1.249736E+03,            1,           0,
  2.710243E-04,   9.224090E-01,   4.829712E-02,   0.000000E+00,   1.249736E+03,            1,           0,
  2.910243E-04,   9.028105E-01,   4.430976E-02,   0.000000E+00,   1.274152E+03,            1,           0,
  2.910243E-04,   9.028105E-01,   4.430976E-02,   0.000000E+00,   1.274152E+03,            1,           0,
  3.110243E-04,   8.832120E-01,   4.032240E-02,   0.000000E+00,   1.260694E+03,            1,           0,
  3.110243E-04,   8.832120E-01,   4.032240E-02,   0.000000E+00,   1.260694E+03,            1,           0,
  3.310243E-04,   8.636136E-01,   3.633504E-02,   0.000000E+00,   1.281517E+03,            1,           0,
  3.310243E-04,   8.636136E-01,   3.633504E-02,   0.000000E+00,   1.281517E+03,            1,           0,
  3.510243E-04,   8.440151E-01,   3.234769E-02,   0.000000E+00,   1.262331E+03,            1,           0,
  3.510243E-04,   8.440151E-01,   3.234769E-02,   0.000000E+00,   1.262331E+03,            1,           0,
  3.710243E-04,   8.244166E-01,   2.836033E-02,   0.000000E+00,   1.269745E+03,            1,           0,
  3.710243E-04,   8.244166E-01,   2.836033E-02,   0.000000E+00,   1.269745E+03,            1,           0,
  3.910243E-04,   8.048181E-01,   2.437297E-02,   0.000000E+00,   1.275913E+03,            1,           0,
  3.910243E-04,   8.048181E-01,   2.437297E-02,   0.000000E+00,   1.275913E+03,            1,           0,
  4.110243E-04,   7.852196E-01,   2.038561E-02,   0.000000E+00,   1.293645E+03,            1,           0,
  4.110243E-04,   7.852196E-01,   2.038561E-02,   0.000000E+00,   1.293645E+03,            1,           0,
  4.310243E-04,   7.656211E-01,   1.639825E-02,   0.000000E+00,   1.322355E+03,            1,           0,
  4.310243E-04,   7.656211E-01,   1.639825E-02,   0.000000E+00,   1.322355E+03,            1,           0,
  4.510243E-04,   7.460226E-01,   1.241089E-02,   0.000000E+00,   1.332978E+03,            1,           0,
  4.510243E-04,   7.460226E-01,   1.241089E-02,   0.000000E+00,   1.332978E+03,            1,           0,
  4.710243E-04,   7.264241E-01,   8.423535E-03,   0.000000E+00,   1.375219E+03,            1,           0,
  4.710243E-04,   7.264241E-01,   8.423535E-03,   0.000000E+00,   1.375219E+03,            1,           0,
  4.910243E-04,   7.068256E-01,   4.436177E-03,   0.000000E+00,   1.396394E+03,            1,           0,
  4.910243E-04,   7.068256E-01,   4.436177E-03,   0.000000E+00,   1.396394E+03,            1,           0,
  5.110243E-04,   6.872271E-01,   4.488181E-04,   0.000000E+00,   1.405183E+03,            1,           0,
  5.110243E-04,   6.872271E-01,   4.488181E-04,   0.000000E+00,   1.405183E+03,            1,           0,
  1.033910E-03,   2.024131E-01,   3.860940E-03,   0.000000E+00,   1.218541E+03,            1,           0,
  1.033910E-03,   2.024131E-01,   3.860940E-03,   0.000000E+00,   1.218541E+03,            1,           0,
  1.053910E-03,   2.220116E-01,   7.848298E-03,   0.000000E+00,   1.281124E+03,            1,           0,
  1.053910E-03,   2.220116E-01,   7.848298E-03,   0.000000E+00,   1.281124E+03,            1,           0,
  1.073910E-03,   2.416101E-01,   1.183566E-02,   0.000000E+00,   1.294444E+03,            1,           0,
  1.073910E-03,   2.416101E-01,   1.183566E-02,   0.000000E+00,   1.294444E+03,            1,           0,
  1.093910E-03,   2.612086E-01,   1.582301E-02,   0.000000E+00,   1.300685E+03,            1,           0,
  1.093910E-03,   2.612086E-01,   1.582301E-02,   0.000000E+00,   1.300685E+03,            1,           0,
```

These preheating results can also be visualized using the visual tool of the user's choice (e.g. Scilab, Matlab, or Python). An example for a similar simulation using a 3 mm × 3 mm block is presented in Figure 10.2.



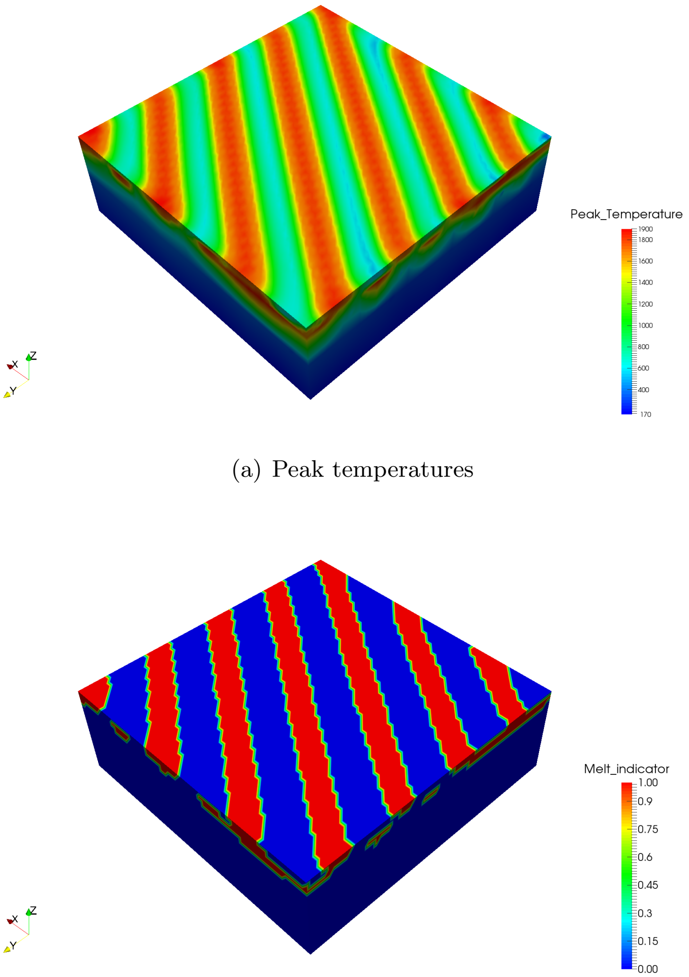
(a) Colorbar

(b) *TPRE Contour

Figure 10.2: Visualization of *TPRE results

10.3.3 Multilayer Thermal Analysis

Figure 10.3 shows the results of the two state variables added to the Paraview results when using *LFUS in the multilayer adaptive simulation. Paraview has a filter option *Threshold* that will only show those elements with values within a specified range for scalar results which makes it easier just to investigate the region of melt. Figure 10.4 shows the peak temperature and melt indicator results for elements which have a melt indicator value of 1.

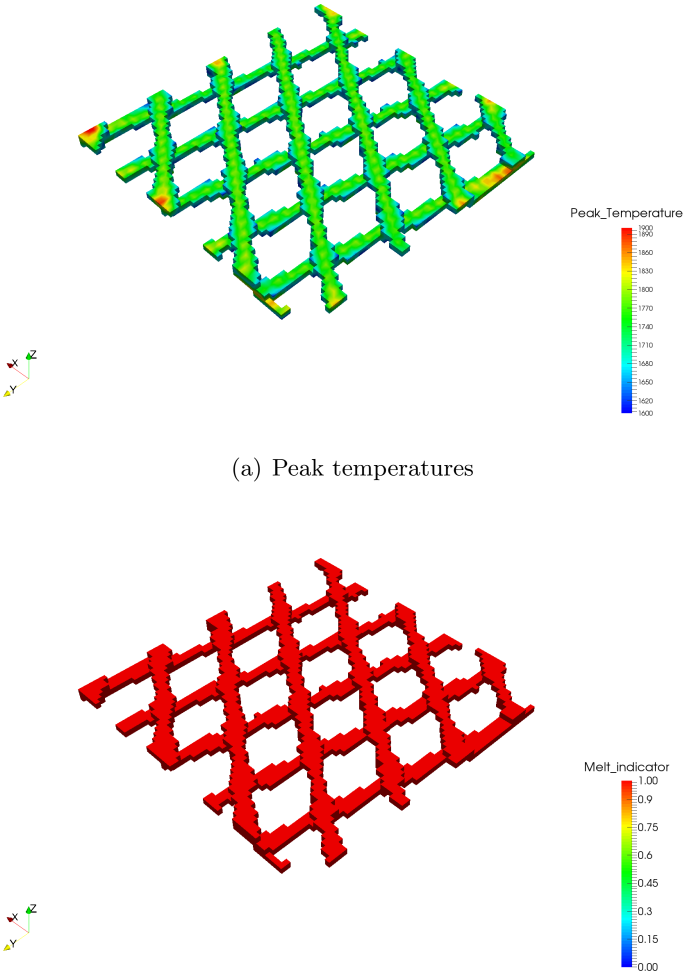


(a) Peak temperatures

(b) Melt indicator

Figure 10.3: LFUS results multilayer mesh

Using *TPRE in this simulation also produces a file `t10multilayer_tpre.txt`, which gives the increment time, heat source location, preheat temperature at that heat source location, and binary flags (1 for those that exceed the test temperatures, 0 for those which fall below) for the temperatures of interest.



(a) Peak temperatures

(b) Melt indicator

Figure 10.4: LFUS results multilayer mesh using threshold filtering

10.3.4 *TPRE results file `t10multilayer_tpre.txt`

These are the results for the first 40 time increments

```
# time, laser_x, laser_y, laser_z, Temp_start, StateVar1 for Temp_crit =   6.900000E+02,   1.600000E+03,
  2.110243E-04,   4.812045E-01,   6.025919E-02,   0.000000E+00,   9.967262E+02,            1,           0,
  2.110243E-04,   4.812045E-01,   6.025919E-02,   0.000000E+00,   9.967262E+02,            1,           0,
  2.310243E-04,   4.616060E-01,   5.627184E-02,   0.000000E+00,   1.047308E+03,            1,           0,
  2.310243E-04,   4.616060E-01,   5.627184E-02,   0.000000E+00,   1.047308E+03,            1,           0,
  2.510243E-04,   4.420075E-01,   5.228448E-02,   0.000000E+00,   1.064964E+03,            1,           0,
  2.510243E-04,   4.420075E-01,   5.228448E-02,   0.000000E+00,   1.064964E+03,            1,           0,
  2.710243E-04,   4.224090E-01,   4.829712E-02,   0.000000E+00,   1.062888E+03,            1,           0,
  2.710243E-04,   4.224090E-01,   4.829712E-02,   0.000000E+00,   1.062888E+03,            1,           0,
  2.910243E-04,   4.028105E-01,   4.430976E-02,   0.000000E+00,   1.068414E+03,            1,           0,
  2.910243E-04,   4.028105E-01,   4.430976E-02,   0.000000E+00,   1.068414E+03,            1,           0,
  3.110243E-04,   3.832120E-01,   4.032240E-02,   0.000000E+00,   1.065790E+03,            1,           0,
  3.110243E-04,   3.832120E-01,   4.032240E-02,   0.000000E+00,   1.065790E+03,            1,           0,
  3.310243E-04,   3.636136E-01,   3.633504E-02,   0.000000E+00,   1.069138E+03,            1,           0,
  3.310243E-04,   3.636136E-01,   3.633504E-02,   0.000000E+00,   1.069138E+03,            1,           0,
  3.510243E-04,   3.440151E-01,   3.234768E-02,   0.000000E+00,   1.070207E+03,            1,           0,
  3.510243E-04,   3.440151E-01,   3.234768E-02,   0.000000E+00,   1.070207E+03,            1,           0,
  3.710243E-04,   3.244166E-01,   2.836032E-02,   0.000000E+00,   1.074631E+03,            1,           0,
  3.710243E-04,   3.244166E-01,   2.836032E-02,   0.000000E+00,   1.074631E+03,            1,           0,
  3.910243E-04,   3.048181E-01,   2.437297E-02,   0.000000E+00,   1.081879E+03,            1,           0,
  3.910243E-04,   3.048181E-01,   2.437297E-02,   0.000000E+00,   1.081879E+03,            1,           0,
  4.110243E-04,   2.852196E-01,   2.038561E-02,   0.000000E+00,   1.089785E+03,            1,           0,
  4.110243E-04,   2.852196E-01,   2.038561E-02,   0.000000E+00,   1.089785E+03,            1,           0,
  4.310243E-04,   2.656211E-01,   1.639825E-02,   0.000000E+00,   1.109395E+03,            1,           0,
  4.310243E-04,   2.656211E-01,   1.639825E-02,   0.000000E+00,   1.109395E+03,            1,           0,
  4.510243E-04,   2.460226E-01,   1.241089E-02,   0.000000E+00,   1.128674E+03,            1,           0,
  4.510243E-04,   2.460226E-01,   1.241089E-02,   0.000000E+00,   1.128674E+03,            1,           0,
  4.710243E-04,   2.264241E-01,   8.423532E-03,   0.000000E+00,   1.158016E+03,            1,           0,
  4.710243E-04,   2.264241E-01,   8.423532E-03,   0.000000E+00,   1.158016E+03,            1,           0,
  4.910243E-04,   2.068256E-01,   4.436173E-03,   0.000000E+00,   1.179002E+03,            1,           0,
  4.910243E-04,   2.068256E-01,   4.436173E-03,   0.000000E+00,   1.179002E+03,            1,           0,
  5.110243E-04,   1.872271E-01,   4.488143E-04,   0.000000E+00,   1.182956E+03,            1,           0,
  5.110243E-04,   1.872271E-01,   4.488143E-04,   0.000000E+00,   1.182956E+03,            1,           0,
  8.739096E-04,   1.879548E-02,   6.822967E-02,   0.000000E+00,   1.008571E+03,            1,           0,
  8.739096E-04,   1.879548E-02,   6.822967E-02,   0.000000E+00,   1.008571E+03,            1,           0,
  8.939096E-04,   3.839398E-02,   7.221703E-02,   0.000000E+00,   1.062931E+03,            1,           0,
  8.939096E-04,   3.839398E-02,   7.221703E-02,   0.000000E+00,   1.062931E+03,            1,           0,
  9.139096E-04,   5.799247E-02,   7.620439E-02,   0.000000E+00,   1.084469E+03,            1,           0,
  9.139096E-04,   5.799247E-02,   7.620439E-02,   0.000000E+00,   1.084469E+03,            1,           0,
  9.339096E-04,   7.759097E-02,   8.019175E-02,   0.000000E+00,   1.089238E+03,            1,           0,
  9.339096E-04,   7.759097E-02,   8.019175E-02,   0.000000E+00,   1.089238E+03,            1,           0,
```

10.3.5 Using timex for multilayer adaptivity simulations

From the command line run:

```
$ timex timex-madap-input.txt
```

This is similar to the timex input file shown in example 1, but with the additional cards ***CRSE** and ***SHFT** which enable timex to work with multilayer adaptivity. The above example timex input file records the temperatures at the center of the build cross section x=0.25 mm, y=0.25 at 6 different z locations. Running the above command will produce the file `timex_peaktemp_t10multilayer.txt`. A plot of the temperatures at 6 queried locations is shown in Figure 10.5.

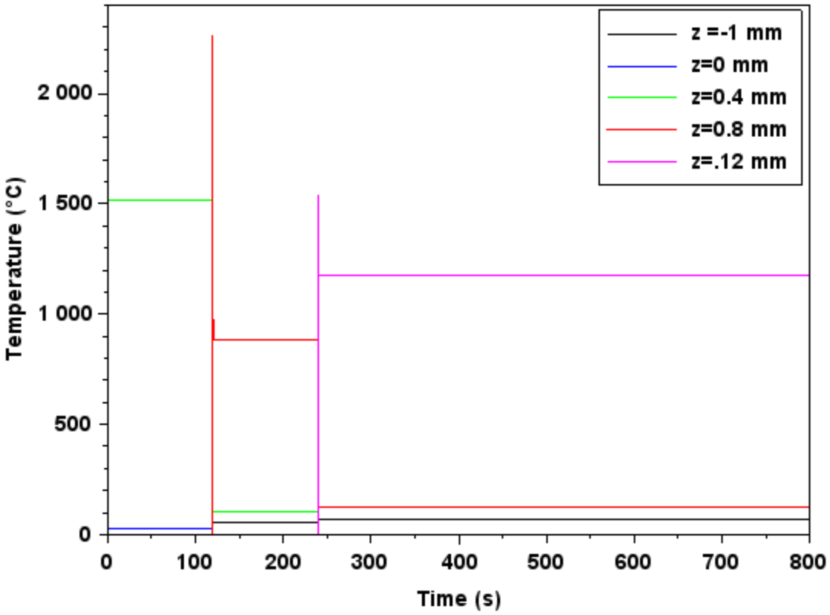


Figure 10.5: Multilayer adaptivity thermal history of the build region center at 6 z locations

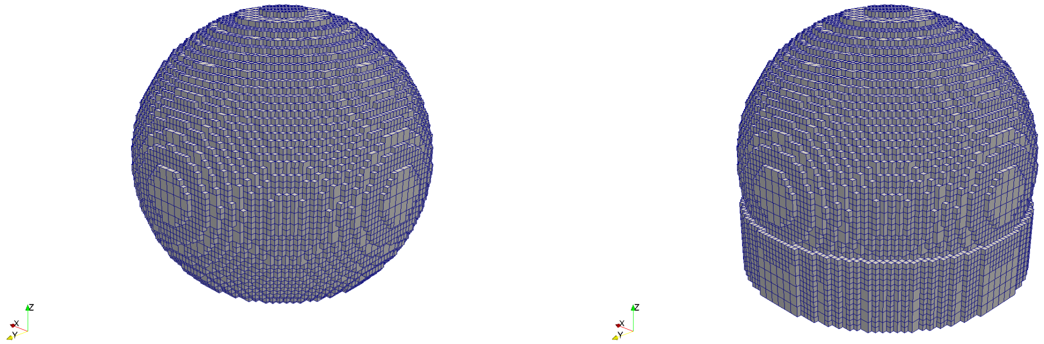
Example 11

Modeling Powder Bed Support Structures using Multiple STLs

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

11.1 Problem Description

An Inconel® 625 spherical geometry with support structures is built in a powder bed system using generic processing parameters. Both the part and support structure geometries are imported in the analysis through STL files and both are automatically meshed within Netfabb Simulation . The buildplate is modeled to be 10.88 mm thick using *DDM!. The time to deposit layers is calculated using the *PBDL card, here modeling the case where 5 identical geometries are built at once. The bottom of the build plate is fixed using the *FSUB card. The *FSUB card will also simulate the release of the buildplate from the machine after the deposition process is complete, but before the part is removed from the buildplate. The mesh, shown with and without the support elements, is shown in Figures 11.1.



(a) Sphere without supports

(b) Sphere with supports

Figure 11.1: Sphere with and without supports.

A time incremental thermal analysis is performed first to compute the temperature history of the part. Layers are activated in groups using *PBPA, and additional time increments are used to model heat conduction into the part. The thermal analysis includes only the part and substrate. Heat loss into the powder is modeled as convection with a value of 25.d-6 W/((mm^2)°C) using *CONV. The build plate is pre-heated to 100 °C.

A time incremental mechanical analysis is performed after the thermal analysis is completed. Similarly to the thermal analysis, layers are activated in groups and the computed temperature distribution from the mechanical analysis is used to compute deformation due to the thermal expansion. These simulations have three additional post-process simulation increments, first Netfabb Simulation simulates the release of the buildplate from the machine, then the removal of the build from the buildplate, and finally the removal of the support structure material from the final build.

11.2 Running Netfabb Simulation

11.2.1 Thermal Analysis

To run the model, from a command line run:

```
$ pan -b t11
```

The analysis progress is written on file multistl_thermal.out. To check progress run:

```
$ tail t11.out
```

After the analysis completes, the last few lines of the output file multistl_thermal.out should be similar to the following:

```
 Increment end
  inc =       38 time =   15820.488     iter =   1 eps =  0.45725E+02
  inc =       38 time =   15820.488     iter =   2 eps =  0.11788E-12
 Finished writing file results/t11_38.case
 Writing record:            2 , time:    15820.4882812500
 Increment end

 Mesh preview volume  =    4480.31249999999
 Activated volume     =    4480.31249999999
 Activated percentage =    100.000000000000

 Finished writing file ./t11.case

 Analysis completed

 CPU wall=   22.91797     CPU total=   79.18611
 Peak RAM used for this process = 190772 kB

 END Netfabb Simulation Solver
```

Actual CPU times will differ.

11.2.2 Quasi-Static Mechanical Analysis

Run the mechanical analysis from the command line:

```
$ pan -b m11
```

After the analysis completes, the last few lines of the output file `multistl_mech.out` should be similar to the following:

```
------------------------------------------
 Support structure removal time increment
------------------------------------------
 inc =       41 time =   19640.488     iter =   1 eps =  0.34691E+03
 inc =       41 time =   19640.488     iter =   2 eps =  0.37281E-09
Finished writing file results/m11_41.case
Increment end

------------------------------------------------------
 Total number of equilibrium iterations:           81

Mesh preview volume  =    4480.31249999999
Activated volume     =    4480.31249999999
Activated percentage =    100.000000000000

Finished writing file ./m11.case

Analysis completed

************************
         12 Warnings
************************

CPU wall=   57.65625     CPU total=   214.5638
Peak RAM used for this process = 807328 kB

END Netfabb Simulation Solver
```

Actual CPU times will differ.

Each of the warnings note a support structure element failure.

11.3 Results

First look at the m11_warning.txt and m11_recoater.txt files to investigate possible build failure. The end of the warning file should look as follows:

```
*** WARNING:           9
Support structure failure
-------------------------------
-------------------------------

*** WARNING:          10
Support structure failure
```

```
-------------------------------
-------------------------------

*** WARNING:           11
Support structure failure
-------------------------------

*** WARNING:           12
Time is too close to previous time for
postprocessing.  Postprocessing will show
artificially increased times from this
increment onwards.

The .out file will still show correct times
```

This indicates some support structure failure will occur. Further investigation of the results will be required to determine the severity of this failure.

```
time (s), layer group, recoater clearance (%), top z deformed coord (mm), recoater coord (mm)
  1.430935E+03           1        90.476  1.188381E+01  1.192000E+01
  2.861970E+03           2        90.871  1.332365E+01  1.336000E+01
  4.293005E+03           3        89.202  1.476432E+01  1.480000E+01
  5.694378E+03           4        85.537  1.620579E+01  1.624000E+01
  7.116843E+03           5        57.511  1.765700E+01  1.768000E+01
  8.547439E+03           6        81.812  1.908728E+01  1.912000E+01
  9.954524E+03           7        86.276  2.052549E+01  2.056000E+01
  1.130953E+04           8        90.012  2.196400E+01  2.200000E+01
  1.258412E+04           9        93.537  2.340259E+01  2.344000E+01
  1.375105E+04          10        96.652  2.484134E+01  2.488000E+01
  1.477691E+04          11        99.985  2.628001E+01  2.632000E+01
  1.563555E+04          12       107.644  2.771694E+01  2.776000E+01
```

This indicates that recoater interference should not be an issue for this geometry. Now look at the simulation results.

Figure 11.2 shows the results for the 3rd to the last increment, before any elements have been removed, after clipping the part to show the center of the build.

These values correspond as follows:
0 - Build plate
1 - Powder
2 - Component
3 - Homogenized component
4 - Support structure
5 - Failed support structure

Observe that there are numerous failed supports, as indicated in both the simulation log file and the m11_warning.txt file. Let us examine the results to ensure these failures did not result in excessive distortion.

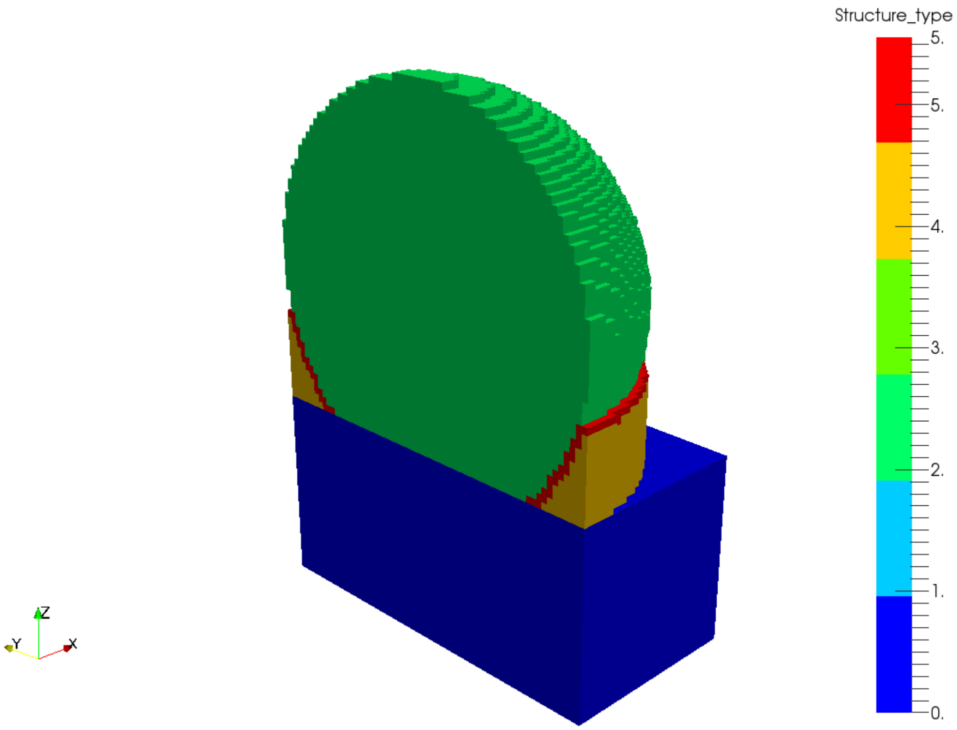
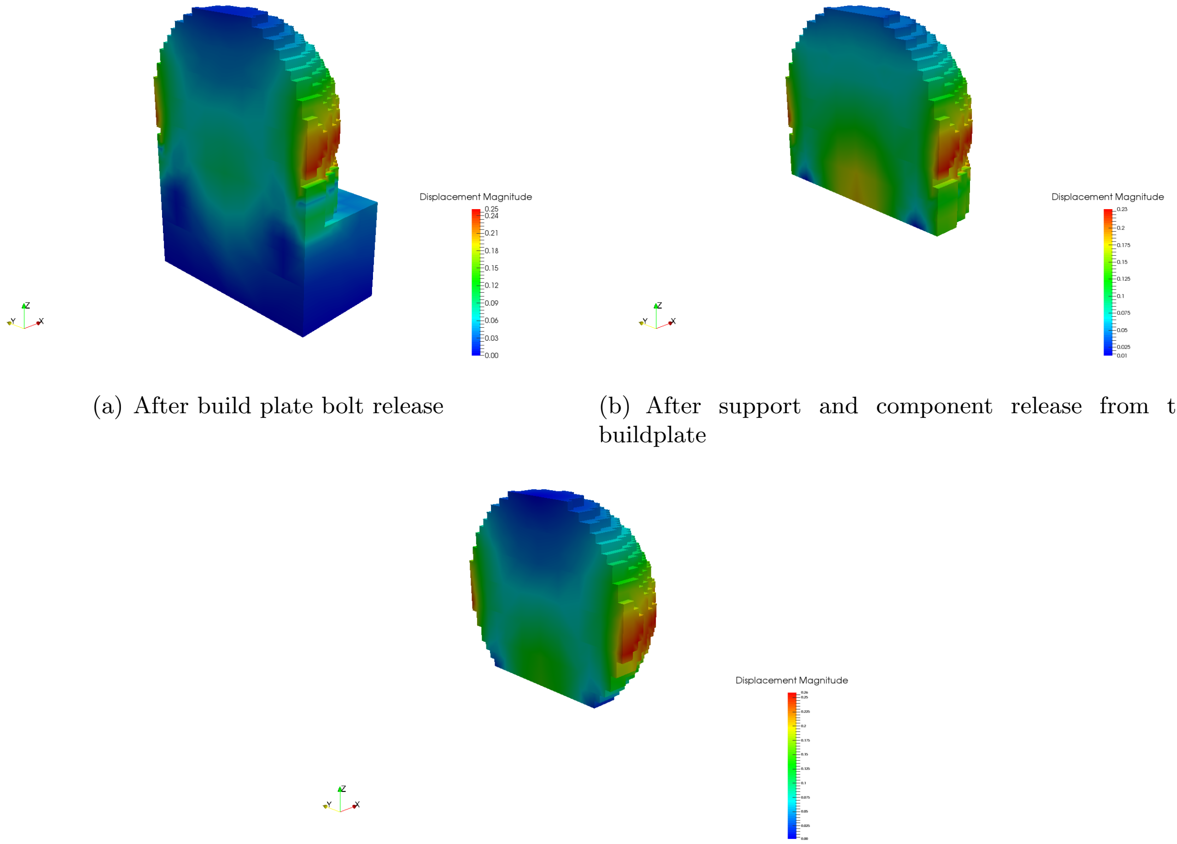


Figure 11.2: Structure type results

Figures 11.3 shows the computed final distortion from the mechanical analysis (m1.in) after the part is removed from the buildplate, and after the support material has been removed.

Observing these results show that despite the support structure failure, the part has maintained a high degree of dimensional accuracy. It is good engineering practice however to check these final dimensions to the tolerances of the part with respect to its end use.



(a) After build plate bolt release

(b) After support and component release from the buildplate

(c) After support material removal from the component

Figure 11.3: Distortion [mm] (1x magnification) basic analysis.

Example 12

Multi-Scale Powder Bed Simulations with Powder

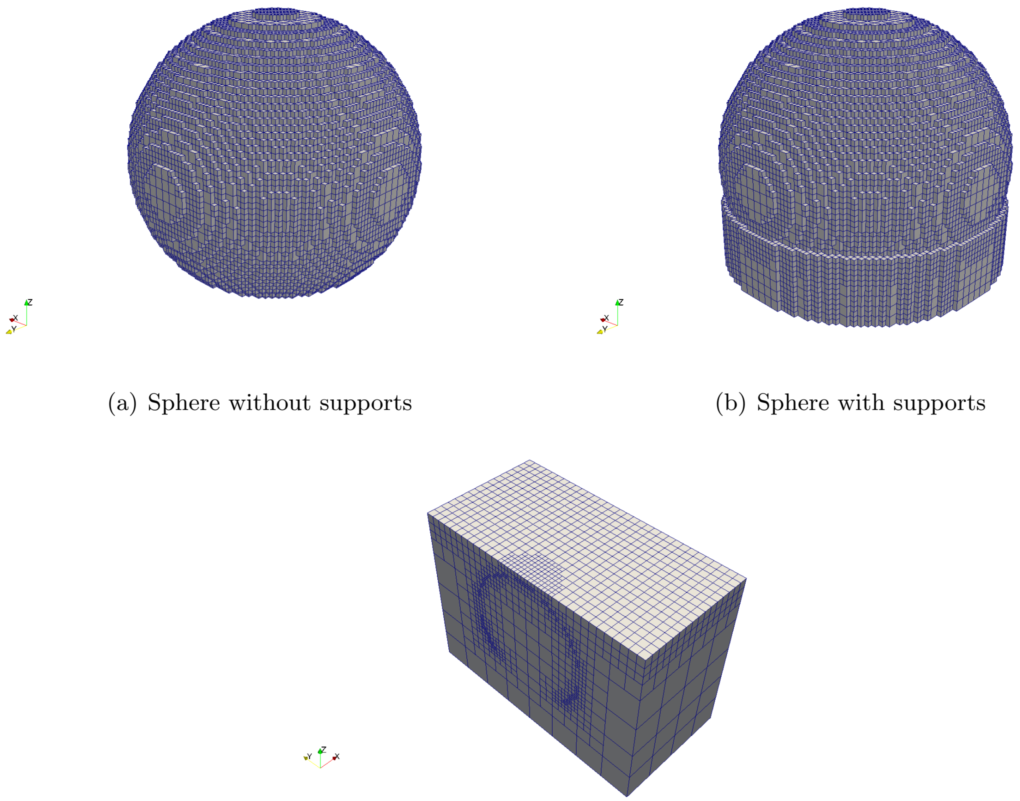
All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

12.1 Problem Description

An Inconel® 625 spherical geometry with support structures is built in a powder bed system using generic processing parameters. Both the part and support structure geometries are imported in the analysis through STL files and both are automatically meshed by the Netfabb Simulation solver. The buildplate is modeled to be 10.88 mm thick using *DDM!. The time to deposit layers is calculated using the *PBDL card, here modeling the case where 5 identical geometries are built at once. The simulation is run twice, once without including powder, and one including powder in the multiscale analysis, using the *+PDR card. Powder properties are automatically scaled. Thermal conductivity of the powder is 0.01× that of the solid while specific heat is 0.6× that of the solid property. The bottom of the build plate is fixed using the *FSUB card. The *FSUB card will also simulate the release of the buildplate from the machine after the deposition process is complete, but before the part is removed from the buildplate. The mesh, shown with support elements, without support elements, and a cross section from the thermal analysis, with meshed powder, is shown in Figures 12.1.

A time incremental thermal analysis is performed first to compute the temperature history of the part. Layers are activated in groups using *PBPA, and additional time increments are used to model heat conduction into the part. The first thermal analysis includes only the part and substrate, with heat loss into the powder being modeled as convection with a value of 25.d-6 W/((mm^2)°C) using *CONV. The second thermal analysis models the powder, part, and substrate. Convection boundary conditions are applied at the surface of the powder and substrate surface, also with a value of 25.d-6 W/((mm^2)°C).

Two time incremental mechanical analyses are performed after the thermal analyses are completed. Similarly to the thermal analyses, layers are activated in groups and the computed temperature distribution from the mechanical analysis is used to compute deformation due to the thermal expansion. These simulations have three additional post-process simulation increments, first Netfabb Simulation simulates the release of the buildplate from the machine, then the removal of the build from the buildplate, and finally the removal of the support structure material from the final build.



(a) Sphere without supports

(b) Sphere with supports

(c) Sectioned sphere with powder elements

Figure 12.1: Sphere with supports, without supports, and with powder elements.

12.2 Running Netfabb Simulation

12.2.1 Thermal Analysis

To run the thermal model without powder, from a command line run:

```
$ pan -b t0
```

The analysis progress is written on file `t0.out`. To check progress run:

```
$ tail t0.out
```

After the analysis completes, the last few lines of the output file `t0.out` should be similar to the following:

```
 Finished writing file results/t0_33.case
 Increment end
  inc =       34 time =   11908.695     iter =   1 eps =  0.15156E+00
  inc =       34 time =   11908.695     iter =   2 eps =  0.72001E-14
 Finished writing file results/t0_34.case
 Increment end
 Finished writing file ./t0.case

 Analysis completed

 CPU wall=   11.60938     CPU total=   229.4411
 END Netfabb Simulation Solver
```

Actual CPU times will differ.

To run the thermal model with powder, from a command line run:

```
$ pan -b t1
```

The analysis progress is written on file `t1.out`. To check progress run:

```
$ tail t1.out
```

After the analysis completes, the last few lines of the output file `t1.out` should be similar to the following:

```
 Finished writing file results/t1_34.case
 Increment end
 Finished writing file ./t1.case

 Analysis completed

 CPU wall=   21.21484     CPU total=   406.9591
 END Netfabb Simulation Solver
```

Actual CPU times will differ.

12.2.2 Quasi-Static Mechanical Analysis

Run the mechanical analysis without powder from the command line:

```
$ pan -b m0
```

After the analysis completes, the last few lines of the output file `m0.out` should be similar to the following:

```
Finished writing file results/m0_37.case
Increment end

---------------------------------------------
 Total number of equilibrium iterations:           73
Finished writing file ./m0.case

Analysis completed

************************
         14 Warnings
************************

CPU wall=   60.24609     CPU total=   1200.464
END Netfabb Simulation Solver
```

Actual CPU times will differ. Each of the warnings note a support structure element failure.

Now run the mechanical analysis with powder from the command line:

```
$ pan -b m1
```

After the analysis completes, the last few lines of the output file `m1.out` should be similar to the following:

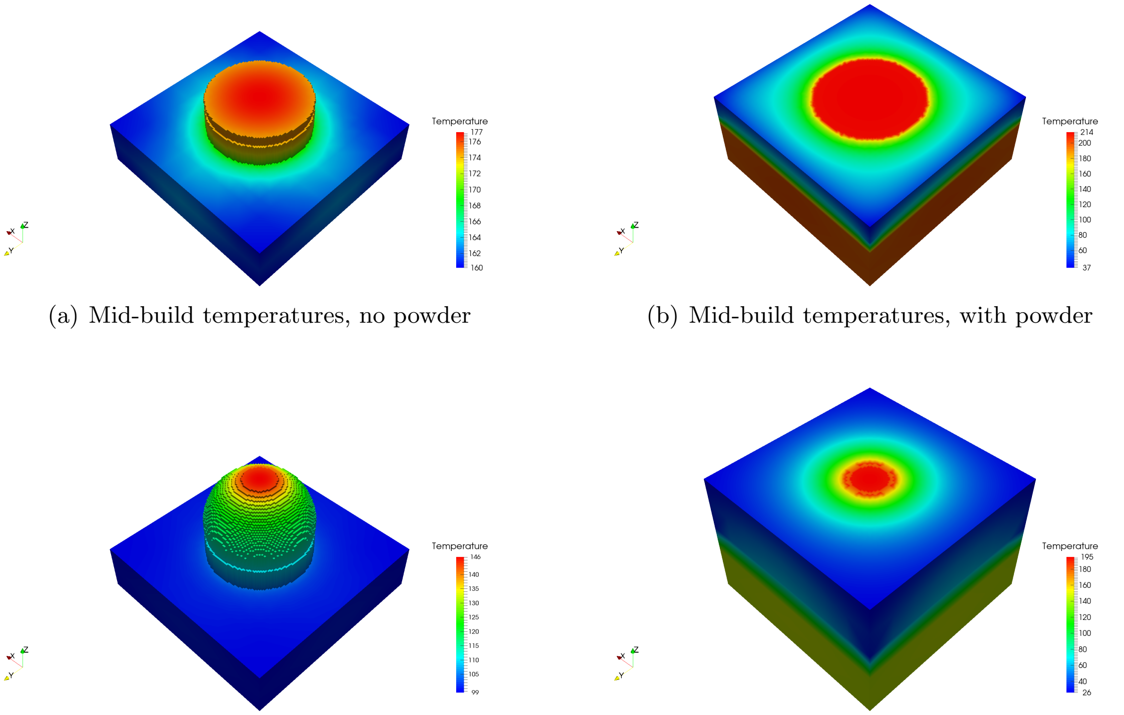
```
Support structure removal time increment
 inc =       37 time =   14908.695     iter =   1 eps =  0.19593E+03
 inc =       37 time =   14908.695     iter =   2 eps =  0.30466E-09
Finished writing file results/m1_37.case
Increment end

---------------------------------------------
 Total number of equilibrium iterations:           73
Finished writing file ./m1.case

Analysis completed

************************
         16 Warnings
************************

CPU wall=   69.24219     CPU total=   1309.613
```

```
 END Netfabb Simulation Solver
```

Actual CPU times will differ. Each of the warnings note a support structure element failure.

12.3 Results

First look at the thermal results. Figure 12.2 shows the model temperature at the end of two different layer group simulations, for both the with and without powder results.



(a) Mid-build temperatures, no powder

(b) Mid-build temperatures, with powder

(c) Near-end-build temperatures, no powder

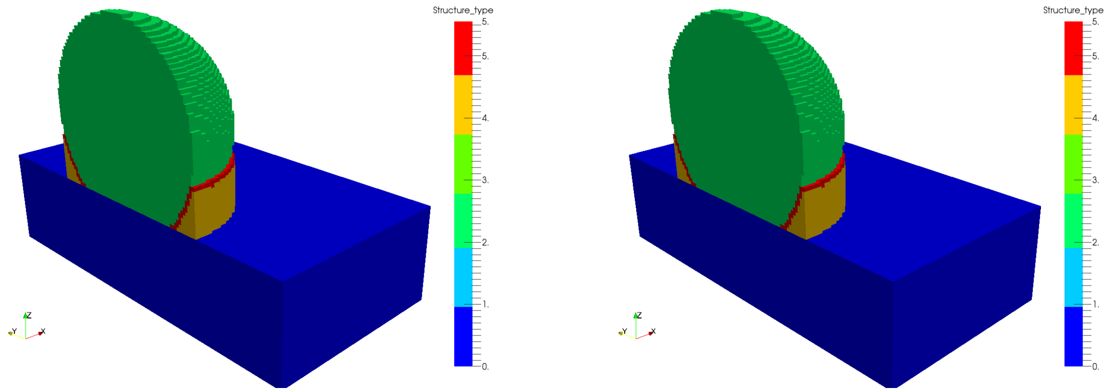
(d) Near-end-build temperatures, with powder

Figure 12.2: Temperatures results at two different time steps for the analysis with and without powder elements included.

Looking at the above temperature it is apparent that including the powder makes a significant difference in the temperature history of the part. The simulations with the powder are warmer than the without powder model, which uses convection to approximate loses due to powder effects. This may have an impact upon the subsequent mechanical simulation results. First look at the support structure failure.

Figure 12.3 shows the results for the 3rd to the last increment, before any elements have been removed, after clipping the part to show the center of the build.

These values correspond as follows:
0 - Build plate
1 - Powder



(a) Structure type, without powder

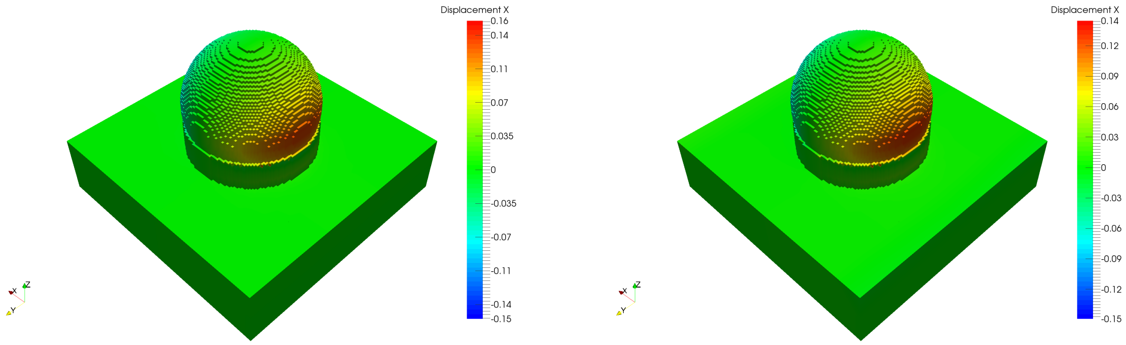
(b) Structure type, with powder

Figure 12.3: Structure type results

2 - Component
3 - Homogenized component
4 - Support structure
5 - Failed support structure

There are failed supports for both simulations. However there were no recoater interference warnings, so these may not have a catastrophic effect. The two simulations predict quite similar, but not identical support structure failures. Most notably including powder shows support failure at the entire part-support junction, while there is a small gap in failure for the simulation without powder. Looking at the displacement results will indicate if these are problematic for production using either approach.

Figures 12.4 shows the predicted displacement in the X direction from the mechanical analysis after build plate release and cooldown, but before support structure or build plate removal.



(a) X displacement, without powder

(b) X displacement, with powder

Figure 12.4: X Displacement, with or without powder [mm] (1x magnification).

Neither of these simulations have excessive distortion. However, note that the simulation with powder distorts 12.5% less than the simulation without powder. This shows that the modeling of heat losses due to powder effects may have a large effect upon the both the predicted distortion.

Example 13

Peak temperature modeling using multi-layer moving adaptivity

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

13.1 Problem Description

This example utilizes the multi-layer moving adaptivity to investigate peak temperatures during Laser Powder Bed Fusion processing. There are two simulations in this example, one which shows peak temperatures only after the final layer, one which shows the peak temperatures develop over the deposition process. Both simulations use a 150 W source moving at 600 mm/s to simulate the melting of two layers of Inconel 718 powder, on a sample 0.5 mm × 0.5 mm × square with 12.5 mm substrate. A surface convection of 10.d-6 W/((mm^2)-°C) is applied on the top surface and the all other faces are insulated. The *ADPM card is used to control the acceptable temperature gradients across an element for coarsening, using the default settings. For both simulations the *LFUS card is used with a value of 1200 °C, to investigate lack of fusion. The mesh and laser path are automatically generated using Netfabb Simulation .

13.2 Running Netfabb Simulation

13.2.1 Final Peak Temperature Model

From a command line run:

```
$ pan -b 13_final
```

The analysis progress is written on file `13_final.out`. To check progress run:

```
$ tail 13_final.out
```

After the analysis completes, the last few lines of the output file `13_final.out` should be similar to the following:

```
Starting refine
Number of refined nodes       =         352
Number of refined elements    =         157
```

```
 Number of equations           =         229
 Number of constrained eqns    =         123

 Starting auxspar
 Number of no zeros nsymmetric =        3989
 Sparse preprocessing complete
  inc =      171 time =   480.00000     iter =   1 eps =  0.15058E-03
  inc =      171 time =   480.00000     iter =   2 eps =  0.10500E-06
 Finished writing file results/13_final_171.case
 Finished writing file results/13_final_171_c.case
 Increment end
 Finished writing file ./13_final.case
 Finished writing file ./13_final_c.case

 Analysis completed

 ************************
           1 Warning
 ************************

 CPU wall=   7.972656     CPU total=   26.42657
 Peak RAM used for this process = 43772 kB

 END Netfabb Simulation Solver
```

Actual CPU times will differ.

13.2.2 Full Peak Temperature History Model

From a command line run:

```
$ pan -b 13_history
```

The analysis progress is written on file `13_history.out`. To check progress run:

```
$ tail 13_history.out
```

After the analysis completes, the last few lines of the output file `13_history.out` should be similar to the following:

```
 Starting refine
 Number of refined nodes       =         352
 Number of refined elements    =         157
 Number of equations           =         229
 Number of constrained eqns    =         123

 Starting auxspar
 Number of no zeros nsymmetric =        3989
 Sparse preprocessing complete
  inc =      171 time =   480.00000     iter =   1 eps =  0.15058E-03
  inc =      171 time =   480.00000     iter =   2 eps =  0.10500E-06
 Finished writing file results/13_history_171.case
 Finished writing file results/13_history_171_c.case
 Increment end
 Finished writing file ./13_history.case
 Finished writing file ./13_history_c.case

 Analysis completed

 ************************
           1 Warning
 ************************

 CPU wall=   9.644531     CPU total=   30.81020
 Peak RAM used for this process = 43136 kB

 END Netfabb Simulation Solver
```

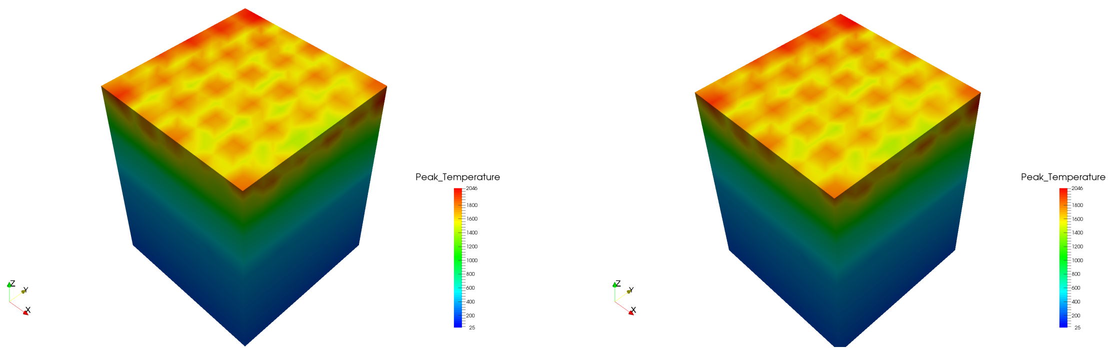
Actual CPU times will differ.

13.3 Results

Results may be imported and viewed in Paraview or Autodesk Simulation Utility for Netfabb.

13.3.1 Peak Temperature results

Figure 13.1 shows the peak temperature results for the final and full history cases, after deposition.



(a) Single step final layer peak temperatures

(b) Multi-layer peak temperatures

Figure 13.1: Peak temperature results

The final results for both cases are identical.

Example 14

Multiscale Thermal Analysis for Lack of Fusion and Hotspot Prediction

14.1 Problem Description

This example illustrates how to generate a Process Parameter file for multi-scale thermal analysis. This analysis tool allows the user to investigate to see how a set of processing parameters will incur lack of fusion or overheating on a particular geometry. This follows the same method in previous chapters, where first a prm file is generated, which simulates the melting of a few layers on a small melt region, then that prm file applied to a part scale model, to predict that geometry will behave if built with the corresponding processing parameters used to generate the prm file.

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

In order to run Part-Level Powder-Bed analysis in Netfabb Simulation , a process parameter (.prm) file must first be generated. The .prm file links the small scale moving-source analysis to the full Part-Level analysis.

To illustrate the usefulness of these options, two prm files are generated in this example, one that results in lack of fusion problems, one that results in hot spots. The lack of fusion processing parameters are:

- Power: 125 W
- Laser spot size: 0.08 mm
- Scan speed: 600 mm/s
- Layer thickness: 0.04 mm
- Hatch spacing: 0.15 mm
- Recoater time: 15 s
- Initial angle rotation: 11.5 degrees
- Interlayer hatch angle rotation: 67 degrees

The hot spot processing parameters are:

- Power: 250 W
- Laser spot size: 0.08 mm
- Scan speed: 600 mm/s
- Layer thickness: 0.04 mm
- Hatch spacing: 0.1 mm
- Recoater time: 15 s
- Initial angle rotation: 11.5 degrees
- Interlayer hatch angle rotation: 67 degrees

The parameters are entered into the *LSRP card. The *GTAB card enables PRM file output and specifies the name of the process parameter file.

14.2 Running Netfabb Simulation

To generate the lack of fusion prm file, from a command line run:

```
$ prm_gen /t 14-fine-lfus.in /i 25 300 600 /l 1290 1350 /o 2200 2600 3000 /d 0.5 0.5 5 > lfus.out
```

The options have the following effects:

/t - Switch for thermal analyses.

/i - Switch for interlayer temperatures. This list, in °C, tabulates the interlayer temperatures to be recorded for subsequent part scale analysis.

/l - Switch for lack of fusion temperatures. This list, in °C, is the temperature or temperatures which may lead to lack of fusion.

/o - Switch for hot spot temperatures. This list, in °C, are for temperature thresholds which may result in deleterious effects.

/d - Switch for dimension control. By default the small scale block for thermal analysis is 1 mm × 1 mm and 5 layers high. This switch overrides those defaults. To decrease the computational time, a smaller block is simulated for this example.

After the prm generation completes, the log file will look as follows:

```
                        15

 prm_gen v1.0.2, Autodesk Inc.

 Thermal PRM input file    = 14-fine-lfus
 Initial temperatures:
   25.0000000000000
   300.000000000000
   600.000000000000
 Lack of fusion temperatures:
   1290.00000000000
   1350.00000000000
 Hot spots temperatures:
   2200.00000000000
   2600.00000000000
   3000.00000000000
 X size =   0.500000000000000
 Y size =   0.500000000000000
 Number of layers =            5

 Reading input file fine-lfus.in
 Generating new table file: LFUS.prm

 Generating threshold fraction data...

 Succesfully generated prm file LFUS.prm
```

This indicates the LFUS.prm file is now available for use in part-scale thermal analyses. But first generate the hot spot example prm file using the same options as the lack of fusion parameter:

```
$ prm_gen /t 14-fine-hotspot.in /i 25 300 600 /l 1290 1350 /o 2200 2600 3000 /d 0.5 0.5 5 > hotspot.out
```

After the prm generation completes, the log file will look as follows:

```
                    15

 prm_gen v1.0.2, Autodesk Inc.

 Thermal PRM input file    = 14-fine-hotspot
 Initial temperatures:
   25.0000000000000
   300.000000000000
   600.000000000000
 Lack of fusion temperatures:
   1290.00000000000
   1350.00000000000
 Hot spots temperatures:
   2200.00000000000
   2600.00000000000
   3000.00000000000
 X size =   0.500000000000000
 Y size =   0.500000000000000
 Number of layers =            5

 Reading input file fine-hotspot.in
 Generating new table file: hotspot.prm

 Generating threshold fraction data...
 Succesfully generated prm file hotspot.prm
```

Now apply these prm files to part scale analyses. First run the part scale lack of fusion example from the command line:

```
$pan -b 14_lfus
```

Use `type` or `tail` to probe the log file, `14_lfus.out`. The end of the log file should look similar to the following:

```
 Finished writing file results/14_lfus_35.case
 Finished writing file results/14_lfus_35_c.case
 Writing record:            2 , time:    3352.03125000000
 Increment end

 Mesh preview volume  =    761.062500000000
 Activated volume     =    761.062500000000
 Activated percentage =    100.000000000000

 Finished writing file ./14_lfus.case
 Finished writing file ./14_lfus_c.case

 Analysis completed

 ************************
           1 Warning
 ************************

 CPU wall=   7.304688     CPU total=   24.75736
 Peak RAM used for this process = 107516 kB

 END Netfabb Simulation Solver
```

Now run the hot spot example:

```
$pan -b 14_hotspot
```

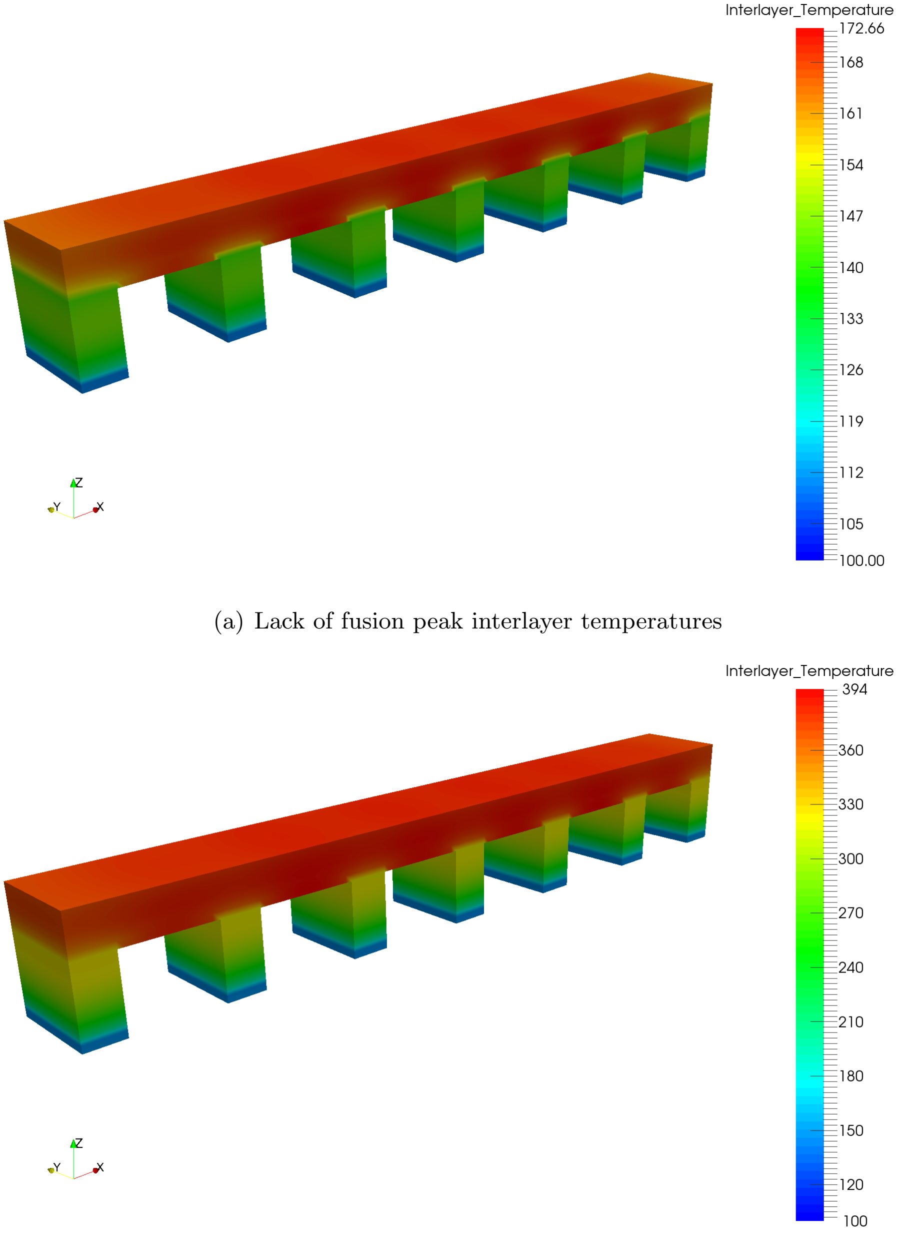
Use `type` or `tail` to probe the log file, `14_hotspot.out`. The end of the log file should look similar to the following:

```
 Finished writing file results/14_hotspot_35.case
 Finished writing file results/14_hotspot_35_c.case
 Writing record:            2 , time:    3880.54687500000
 Increment end

 Mesh preview volume  =    761.062500000000
 Activated volume     =    761.062500000000
 Activated percentage =    100.000000000000

 Finished writing file ./14_hotspot.case
 Finished writing file ./14_hotspot_c.case

 Analysis completed
```

```
************************
          1 Warning
************************

CPU wall=   7.289062     CPU total=   23.72775
Peak RAM used for this process = 108496 kB

END Netfabb Simulation Solver
```

14.3 Results

Figure 14.1 shows the peak interlayer temperatures from the two part scale simulations. The temperatures displayed are those after each layer of material is melted and the cool down period has completed. Observe how the hotspot case has much higher temperatures than the lack of fusion case. This will impact both the lack of fusion and hotspot behavior in each simulation, presented later.

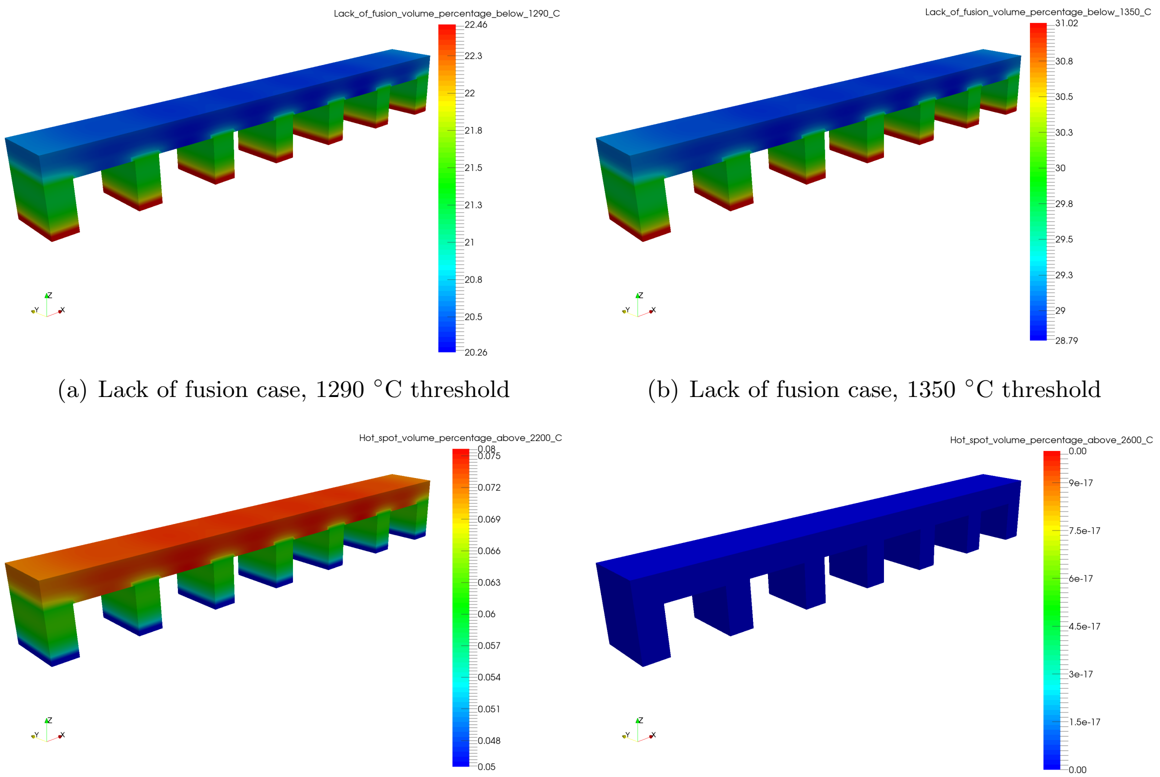


(a) Lack of fusion peak interlayer temperatures

(b) Hotspot peak interlayer temperatures

Figure 14.1: Peak temperature results for the two cases, in °C

Figure 14.2 presents the lack of fusion case threshold results.



(a) Lack of fusion case, 1290 °C threshold

(b) Lack of fusion case, 1350 °C threshold

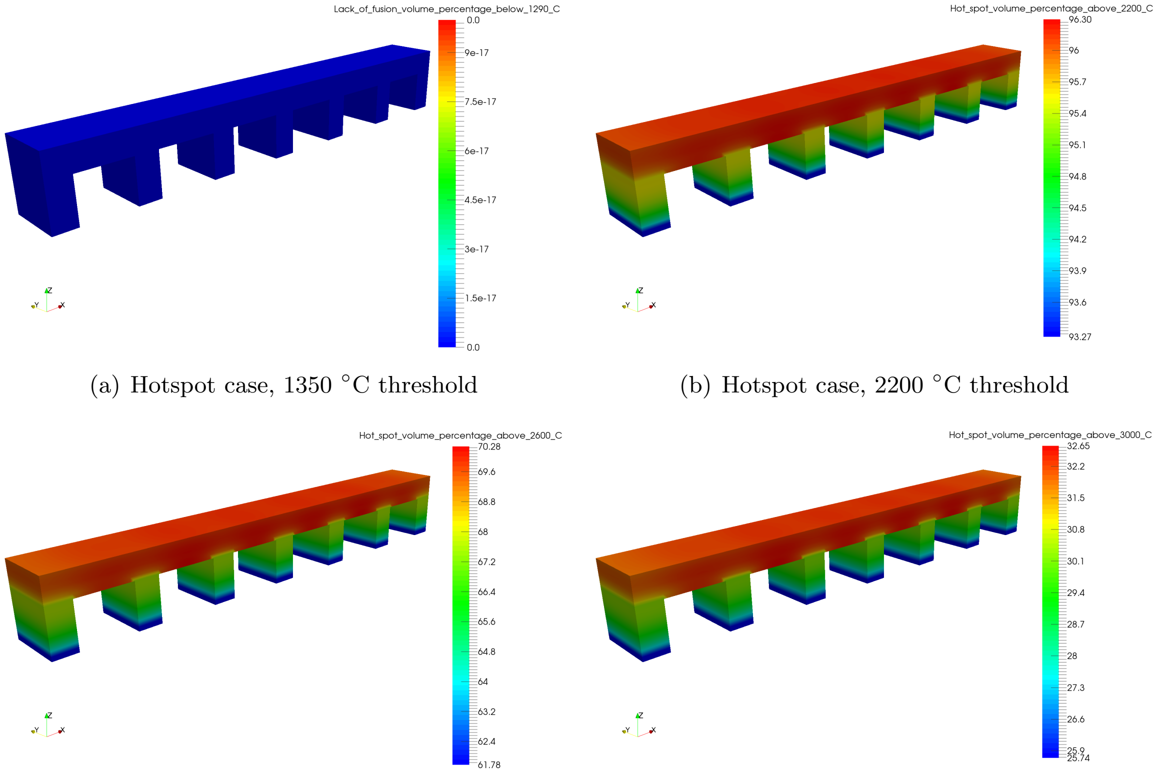
(c) Lack of fusion case, 2200 °C threshold

(d) Lack of fusion case, 2600 °C threshold

Figure 14.2: Lack of fusion case results

For this case, which intentionally used low heat input processing parameters, there is clear indication that there is lack of fusion problems. The minimum lack of fusion volume percentage for the 1290°C temperature is 20.3%. This indicates that more than 20% of the deposited volume does not even reach the solidus temperature, indicating there will be significant lack of fusion. The 1350°C check shows that over 28% of the build volume does not fully melt. This could cause lack of adhesion or build failure early on in the production of this part. Overheating is not an issue with this set of processing conditions, as a negligible portion of the build volume exceeds 2200°C, and none of the part reaches 2600°C.

Figure 14.3 presents the hotspot case results.



(a) Hotspot case, 1350 °C threshold

(b) Hotspot case, 2200 °C threshold

(c) Hotspot case, 2600 °C threshold

(d) Hotspot case, 3000 °C threshold

Figure 14.3: Hotspot case results

As expected the hotspot results are opposite of the lack of fusion case. The part shows no lack of fusion problems, with all of the deposited part reaching the liquidus temperature of 1350°C. More than 93% of the part reaches at least 2200°C, more than 61% of the volume exceeds 2600°C, and more than 25% of the volume is heated oever 3000°C. This shows these processing parameters create excessive heat. Hotspot reduction may be achieved by reducing the power input, increasing the interlayer dwell time, or adding support structures to draw heat away from the unconnected regions into the base plate.

Example 15

Thermo-mechanical processing and heat treatment modeling

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

15.1 Problem Description

A simulation of a laser powder bed fusion build of a generic geometry from Inconel®625 on a SAE 304 build plate is completed using generic processing conditions, then the heat treatment of the component is modeled. The substrate is assumed to be 25mm thick. The resulting mesh is illustrated in Figures 16.1.

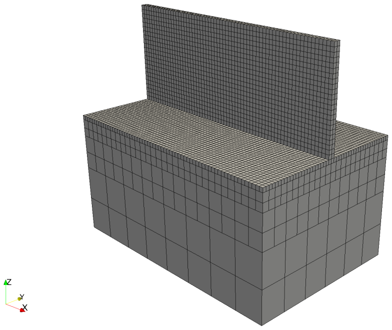


Figure 15.1: Autogenerated mesh.

A time incremental thermal analysis is performed first to compute the temperature history of the part. Layers are activated in groups, and additional time increments are used to model heat conduction into the part. The thermal analysis includes only the part and substrate. Heat loss into the powder is modeled as convection with a value of 25.d-6 W/((mm^2)°C) using the *CONV option.

A time incremental mechanical analysis is performed after the thermal analysis is completed using quantitative stress analysis settings. Similarly to the thermal analysis, layers are activated in groups using *PBPA and the computed temperature distribution from the mechanical analysis is used to compute deformation due to the thermal expansion.

At the end of the build simulation heat treatment of the component and build plate is modeled using a sample heat treatment schedule to stress relieve the part. The build plate is heated to 899°C over a half an hour, held at 899°C for 2.5 hours, and then cooled down to ambient temperature over 3 hours. The aim of the stress relieving temperature is to remove around 90% of the residual stresses of the as built part.

15.2 Running Netfabb Simulation

15.2.1 Thermal Analysis

To run the model, from a command line run:

```
$ pan -b ht_bench_t
```

The analysis progress is written on file `ht_bench_t.out`. To check progress run:

```
$ tail ht_bench_t.out
```

After the analysis completes, the last few lines of the output file `ht_bench_t.out` should be similar to the following:

```
Heat treatment step #  6
 Heat treatment time =  21600.000
  inc =       60 time =   49984.688     iter =    1 eps =  0.32805E+01
  inc =       60 time =   49984.688     iter =    2 eps =  0.12222E+00
  inc =       60 time =   49984.688     iter =    3 eps =  0.23352E-02
 Finished writing file results/ht_bench_t_60.case
 Writing record:            6 , time:    49984.6875000000
 Increment end

 Mesh preview volume  =    9347.00000000000
 Activated volume     =    9347.00000000000
 Activated percentage =    100.000000000000

 Finished writing file ./ht_bench_t.case

 Analysis completed

 CPU wall=   13.07422     CPU total=   38.45425
 Peak RAM used for this process = 120648 kB

 END Netfabb Simulation Solver
```

Actual CPU times will differ.

15.2.2 Quasi-Static Mechanical Analysis

Run the mechanical analysis from the command line:

```
$ pan -b ht_bench_m
```

The analysis progress is written in the file `ht_bench_m.out`. To check progress run:

```
$ tail ht_bench_m.out
```

After the analysis completes, the last few lines of the output file `ht_bench_m.out` should be similar to the following:

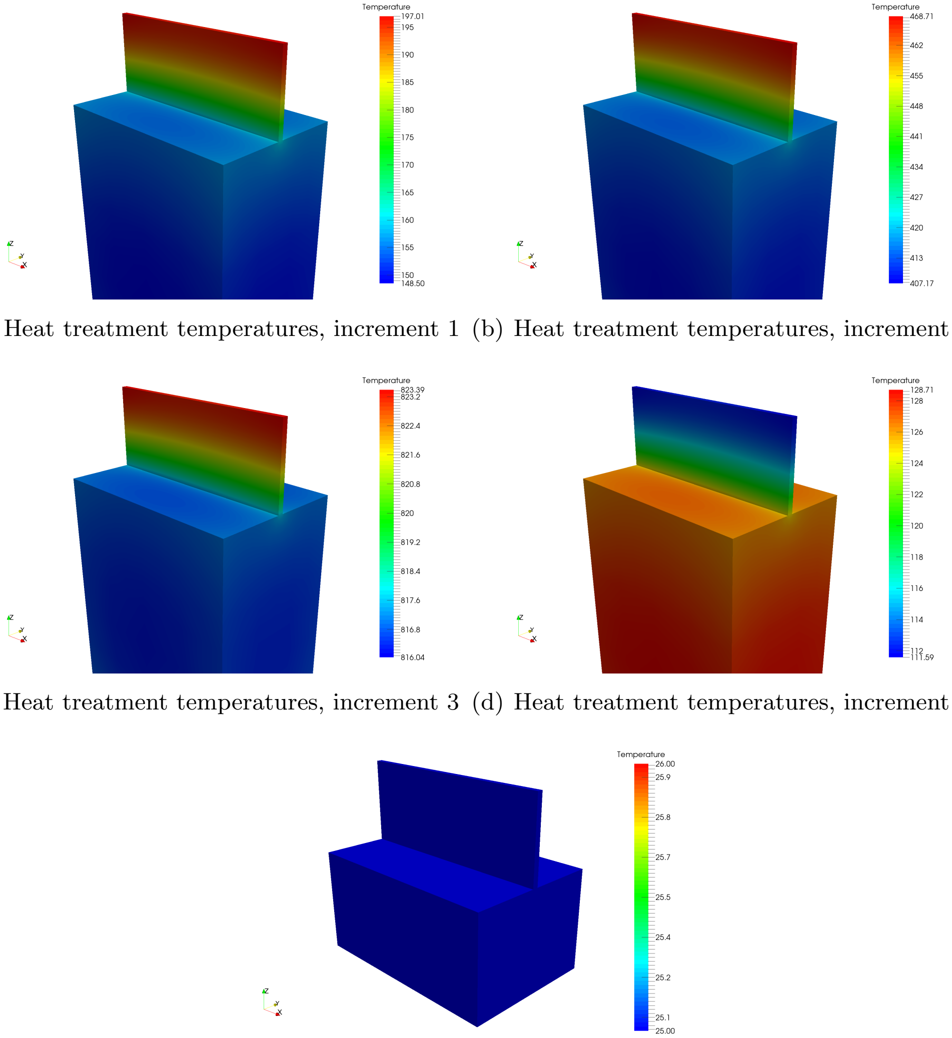
```
Heat treatment step #  6
 Heat treatment time =  21600.000
  inc =       62 time =   199724.69     iter =   1 eps =  0.20354E+07
  inc =       62 time =   199724.69     iter =   2 eps =  0.39207E-08
 Finished writing file results/ht_bench_m_62.case
 Increment end

 ----------------------------------
  Substrate removal time increment
 ----------------------------------
  inc =       63 time =   249724.69     iter =   1 eps =  0.82766E+05
  inc =       63 time =   249724.69     iter =   2 eps =  0.66267E-09
 Finished writing file results/ht_bench_m_63.case
 Increment end

 ------------------------------------------------------
  Total number of equilibrium iterations:          143

 Mesh preview volume  =    9347.00000000000
 Activated volume     =    9347.00000000000
 Activated percentage =    100.000000000000

 Finished writing file ./ht_bench_m.case

 Analysis completed

 ************************
           1 Warning
 ************************

 CPU wall=   35.59766     CPU total=   109.2631
 Peak RAM used for this process = 334168 kB

 END Netfabb Simulation Solver
```

Actual CPU times may differ. Note both the plasticity and heat treatment steps after the build simulation finishes.

15.3 Results

Results may be imported and viewed in Paraview or Simulation Utility for Netfabb.

Figures 15.2 shows the temperatures during the simulated heat treatment.



(a) Heat treatment temperatures, increment 1 (b) Heat treatment temperatures, increment 2

(c) Heat treatment temperatures, increment 3 (d) Heat treatment temperatures, increment 4

(e) Heat treatment temperatures, increment 5

Figure 15.2: Heat treatment temperature results, temperatures in °C

The modeled stresses in the component and build plate after the plasticity model is introduced but before heat treatment is shown in Figure 15.3. These are the high residual stresses which make using non-heat treated AM parts inadvisable.

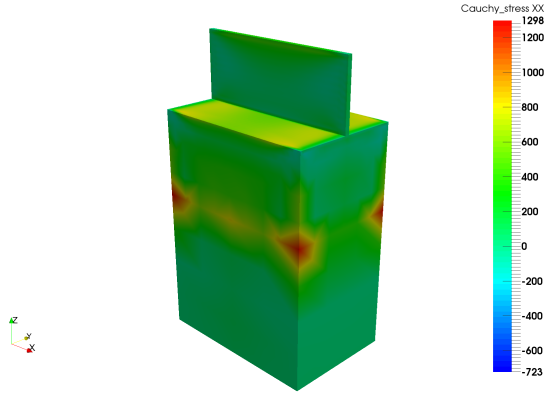
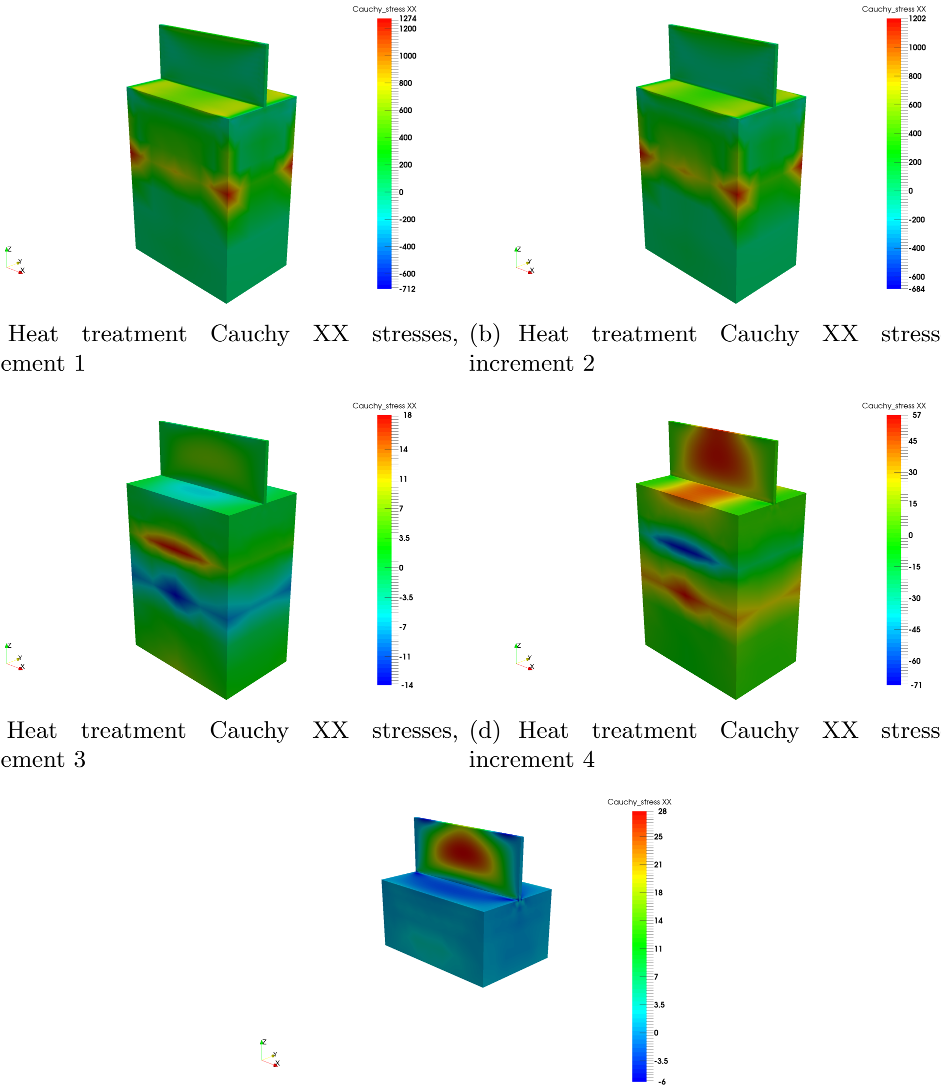


Figure 15.3: Modeled stresses prior to heat treatment

Figures 15.4 shows the modeled stresses during the heat treatment simulation.



(a) Heat treatment Cauchy XX stresses, increment 1

(b) Heat treatment Cauchy XX stresses, increment 2

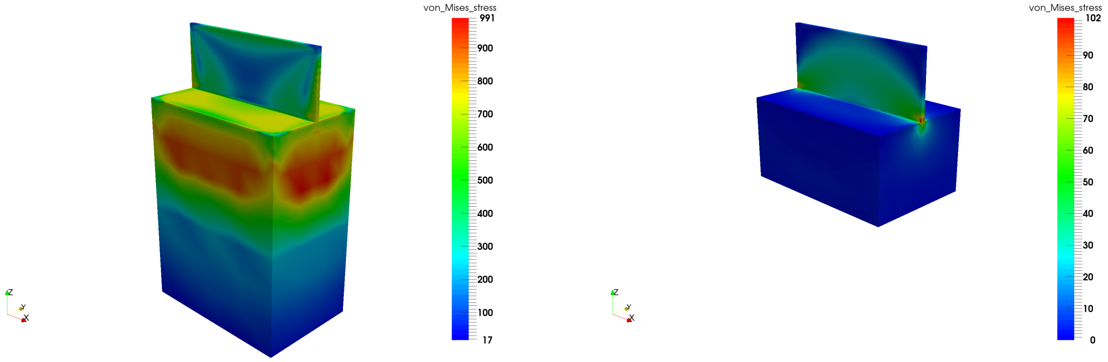
(c) Heat treatment Cauchy XX stresses, increment 3

(d) Heat treatment Cauchy XX stresses, increment 4

(e) Heat treatment Cauchy XX stresses, increment 5

Figure 15.4: Heat treatment Cauchy XX stress results

Observe the stresses reduce as the temperatures increase. Then as the heated part is slowly brought back to room temperature, new thermal stresses are introduced. Looking at the Von Mises Stresses in Figure 15.5, the final, room temperature part has peak stresses that are 90% of the peak stresses prior to heat treatment. This achieves the goal of removing 90% of the stresses induced by the LPBF construction process by the means of stress relief heat treatment.



(a) Heat treatment Von Mises stresses, before heat treatment

(b) Heat treatment Von Mises stresses, after heat treatment

Figure 15.5: Von Mises stress results before and after heat treatment

Example 16

Heat treatment modeling using the restart capabilities

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

16.1 Problem Description

A simulation of a laser powder bed fusion build of a generic geometry from Inconel®625 on a SAE 304 build plate is completed using generic processing conditions, then the heat treatment of the component is modeled using two different heat treatment cycles. The substrate is assumed to be 25mm thick. The resulting mesh is illustrated in Figures 16.1.

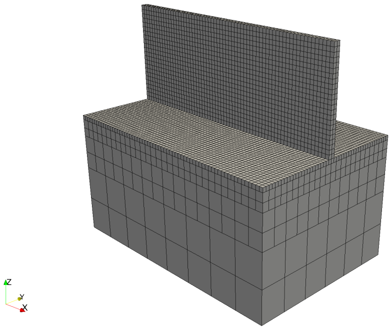


Figure 16.1: Autogenerated mesh.

A time incremental thermal analysis is performed first to compute the temperature history of the part. Layers are activated in groups, and additional time increments are used to model heat conduction into the part. The thermal analysis includes only the part and substrate. Heat loss into the powder is modeled as convection with a value of 25.d-6 W/((mm^2)°C) using the *CONV option.

A time incremental mechanical analysis is performed after the thermal analysis is completed using quantitative stress analysis settings. Similarly to the thermal analysis, layers are activated in groups using *PBPA and the computed temperature distribution from the mechanical analysis is used to compute deformation due to the thermal expansion.

At the end of the build simulation heat treatment of the component and build plate is modeled using a sample heat treatment schedule to stress relieve the part. The build plate is heated to 899°C over a half an hour, held at 899°C for 2.5 hours, and then cooled down to ambient temperature over 3 hours. The aim of the stress relieving temperature is to remove around 90% of the residual stresses of the as built part. During this simulation, restart files are written using the *ORES control card.

A second heat treatment cycle is modeled using the exact same model and processing conditions. Using the *REST card, this can be achieved without having to rerun the process simulation. The heat treatment cycle begins at the end of the processing model, using the previous model results. The new heat treatment cycle heats the chamber to 700°C over the course of 2 hours, is held at a constant temperature for just under 4 hours, and cooled to room temperature in almost 6 hours.

16.2 Running Netfabb Simulation

16.2.1 Thermal Analysis

To run the original thermal model, from a command line run:

```
$ pan -b ht_bench_t
```

After the analysis completes, the last few lines of the output file `ht_bench_t.out` should be similar to the following:

```
Heat treatment step #  6
 Heat treatment time =  21600.000
  inc =       60 time =   49984.688     iter =   1 eps =  0.32805E+01
  inc =       60 time =   49984.688     iter =   2 eps =  0.12222E+00
  inc =       60 time =   49984.688     iter =   3 eps =  0.23352E-02
 Finished writing file results/ht_bench_t_60.case
 Writing record:            6 , time:    49984.6875000000
 Increment end

 Mesh preview volume  =    9347.00000000000
 Activated volume     =    9347.00000000000
 Activated percentage =    100.000000000000

 Finished writing file ./ht_bench_t.case

 Analysis completed

 CPU wall=   13.07422     CPU total=   38.45425
 Peak RAM used for this process = 120648 kB

 END Netfabb Simulation Solver
```

Actual CPU times will differ.

16.2.2 Quasi-Static Mechanical Analysis

Run the original mechanical analysis from the command line:

```
$ pan -b ht_bench_m
```

After the analysis completes, the last few lines of the output file `ht_bench_mechanical.out` should be similar to the following:

```
Heat treatment step #  6
 Heat treatment time =  21600.000
  inc =       62 time =   199724.69     iter =   1 eps =  0.20354E+07
  inc =       62 time =   199724.69     iter =   2 eps =  0.39207E-08
 Finished writing file results/ht_bench_m_62.case
 Increment end

 ----------------------------------
  Substrate removal time increment
 ----------------------------------
  inc =       63 time =   249724.69     iter =   1 eps =  0.82766E+05
  inc =       63 time =   249724.69     iter =   2 eps =  0.66267E-09
 Finished writing file results/ht_bench_m_63.case
 Increment end

 ------------------------------------------------------
  Total number of equilibrium iterations:          143

 Mesh preview volume  =    9347.00000000000
 Activated volume     =    9347.00000000000
 Activated percentage =    100.000000000000

 Finished writing file ./ht_bench_m.case

 Analysis completed

 ************************
           1 Warning
 ************************

 CPU wall=   35.59766     CPU total=   109.2631
 Peak RAM used for this process = 334168 kB

 END Netfabb Simulation Solver
```

Actual CPU times may differ. Now the restart simulations are performed. First the thermal model is rerun

```
$ pan -b ht_bench_t_restart
```

After the analysis completes, the last few lines of the output file `ht_bench_t_restart.out` should be similar to the following:

```
 Heat treatment step #  5
```

```
Heat treatment time =  41600.000
Furnace temperature =  25.000000
 inc =       57 time =   69724.688     iter =   1 eps =  0.26334E+01
 inc =       57 time =   69724.688     iter =   2 eps =  0.42881E-01
 inc =       57 time =   69724.688     iter =   3 eps =  0.46977E-03
Finished writing file results/ht_bench_t_restart_57.case
Writing record:            5 , time:    69724.6875000000
Increment end
Layer end

Mesh preview volume  =    9347.00000000000
Activated volume     =    9347.00000000000
Activated percentage =    100.000000000000

Finished writing file ./ht_bench_t_restart.case

Analysis completed

************************
          1 Warning
************************

CPU wall=   3.890625     CPU total=   10.03086
Peak RAM used for this process = 119160 kB

END Netfabb Simulation Solver
```

Actual CPU times will differ but note the restart Wall time is about 1/3 of the original thermal simulation Wall time. Now run the new heat treatment mechanical simulation.

```
   $ pan -b ht_bench_m_restart
```

After the analysis completes, the last few lines of the output file `ht_bench_mechanical.out` should be similar to the following:

```
Heat treatment time =  41600.000
Furnace temperature =  25.000000
 inc =       60 time =   219204.69     iter =   1 eps =  0.17748E+07
 inc =       60 time =   219204.69     iter =   2 eps =  0.28468E+03
 inc =       60 time =   219204.69     iter =   3 eps =  0.61229E+00
 inc =       60 time =   219204.69     iter =   4 eps =  0.50482E-05
Finished writing file results/ht_bench_m_restart_60.case
Increment end
Layer end

----------------------------------
 Substrate removal time increment
----------------------------------
```

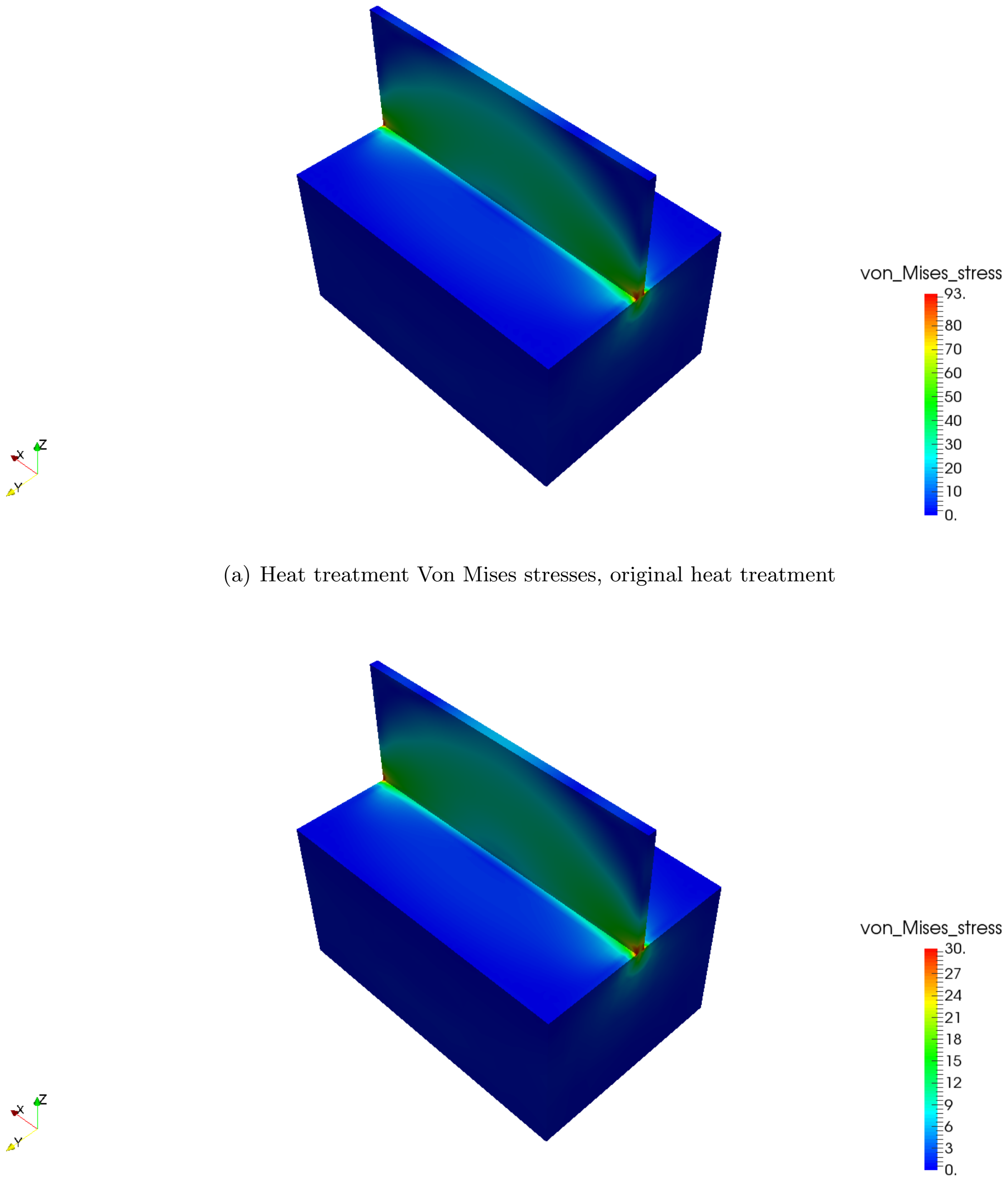
```
 inc =       61 time =   269204.69     iter =    1 eps =  0.43481E+05
 inc =       61 time =   269204.69     iter =    2 eps =  0.39146E-09
Finished writing file results/ht_bench_m_restart_61.case
Increment end

------------------------------------------------------
 Total number of equilibrium iterations:          39

Mesh preview volume  =    9347.00000000000
Activated volume     =    9347.00000000000
Activated percentage =    100.000000000000

Finished writing file ./ht_bench_m_restart.case

Analysis completed

************************
          1 Warning
************************

CPU wall=   12.57031     CPU total=   44.83469
Peak RAM used for this process = 368016 kB

END Netfabb Simulation Solver
```

Actual simulation times will differ, but again note the simulation time using the restarted input file is about 1/3 of the original simulation.

16.3 Results

Results may be imported and viewed in Paraview or Simulation Utility for Netfabb.

Figure 16.2 displays the model results of the original and new heat treatment cycle simulations. The simulation results of the original case show a reduction of 90%, as shown in the previous example 15. Using restart files, the new heat treatment cycle which has slower heat up and cool down periods and lower temperatures, the resulting peak stresses are about 1/3 of those of the original heat treatment cycle. This illustrates the usefulness of the restart capability to optimize heat treatment schedules using Netfabb Simulation .



(a) Heat treatment Von Mises stresses, original heat treatment

(b) Heat treatment Von Mises stresses, new heat treatment

Figure 16.2: Von Mises stress results using the original and new heat treatment cycles

Example 17

Thermal modeling using advanced convection boundary conditions

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

17.1 Problem Description

The current example illustrates the use of advanced convection boundary conditions to create more accurate thermal models without necessitating the use of trapped powder elements. Four simulations comprise this example: a powder element case, a case using the global convection boundary, and two simulations using advanced convection boundary condition options. All simulations use the same geometry, Inconel 625 PRM file, build plate material properties, and mesh settings.

17.2 Running Netfabb Simulation

17.2.1 Powder element simulation

First, a simulation using powder elements should be run to which to compare the convection approximations.

From a command line run:

```
$ pan -b pdr
```

The analysis progress is written on file `pdr.out`.

After the analysis completes, make note of the CPU wall time from the log file. This case takes 52 seconds to complete. Actual CPU times will differ.

Now run the global convection model, which approximates losses into the powder and ambient environment as a uniform heat flux of 25 W/mm^2 K.

```
$ pan -b global
```

The log file returns a CPU time of 19 seconds for this simulation. CPU times may vary.

Next run the 1st advanced convection approximation model which applies regional convection values. These values are:

- Global Convection - *CONV = 5 W/mm^2 K. This low convection value is used to approximate losses from the sides of the part into poorly conductive powder.
- Powder Bed Top Convection - *PBCT = 5 W/mm^2 K. *PBCT are summed with the *CONV value, creating a total convective loss at the top of the part of 10 W/mm^2 K. This accounts for additional convective losses due to natural convection and the forced convection caused by the gas flow over the deposition surface.
- Powder Bed Substrate convection - *PBSB = 250 W/mm^2 K. This approximates losses from the build plate base into the build elevator. A high convection boundary is necessary to adequately model conduction heat losses as a heat flux.
- Powder Bed Substrate Sides convection - *PCSS = 150 W/mm^2 K. This approximate losses from the build plate sides into the powder and walls of the powder bed machine. This is also an applied heat flux simulating conduction losses, but as there is a thin layer of powder between the solid build plate and the solid machine walls, the rate of heat transfer is less than for the build plate-build elevator surface.

```
$ pan -b regional
```

The regional log file shows a CPU time of 18 seconds. CPU times may vary.

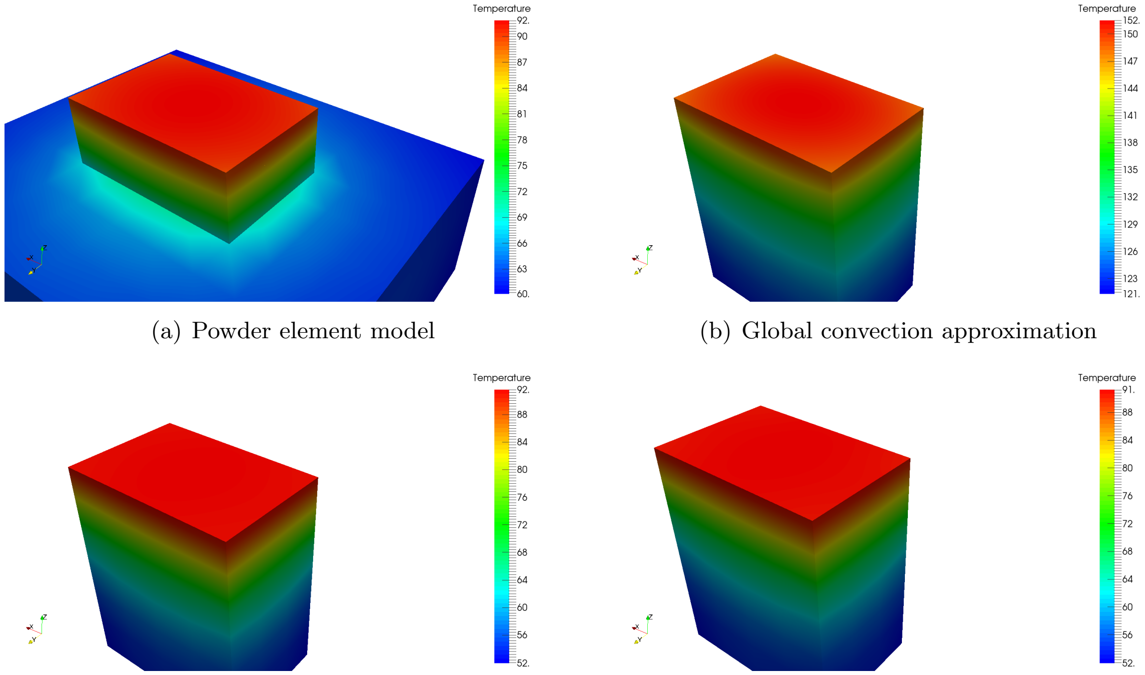
Finally, run the 2nd advanced convection approximation model which uses the same values as the regional case, but has an additional control card *TCNV. This card assigns different convection values based upon the thickness of the component. These values override the values specified by *CONV. For this example the thick sections will be given a flux of 5 W/mm^2 K while the thin sections, which loose heat more rapidly, will have a heat flux of 20 W/mm^2 K.

```
$ pan -b thickness
```

The thickness simulation also take 18 seconds to complete.

17.2.2 Thermal results

Figure 17.1 the thermal results at increment 20, where the thick base of the part is being modeled. Powder simulation results have had the powder elements removed for easier comparison with the non-powder cases.



(a) Powder element model

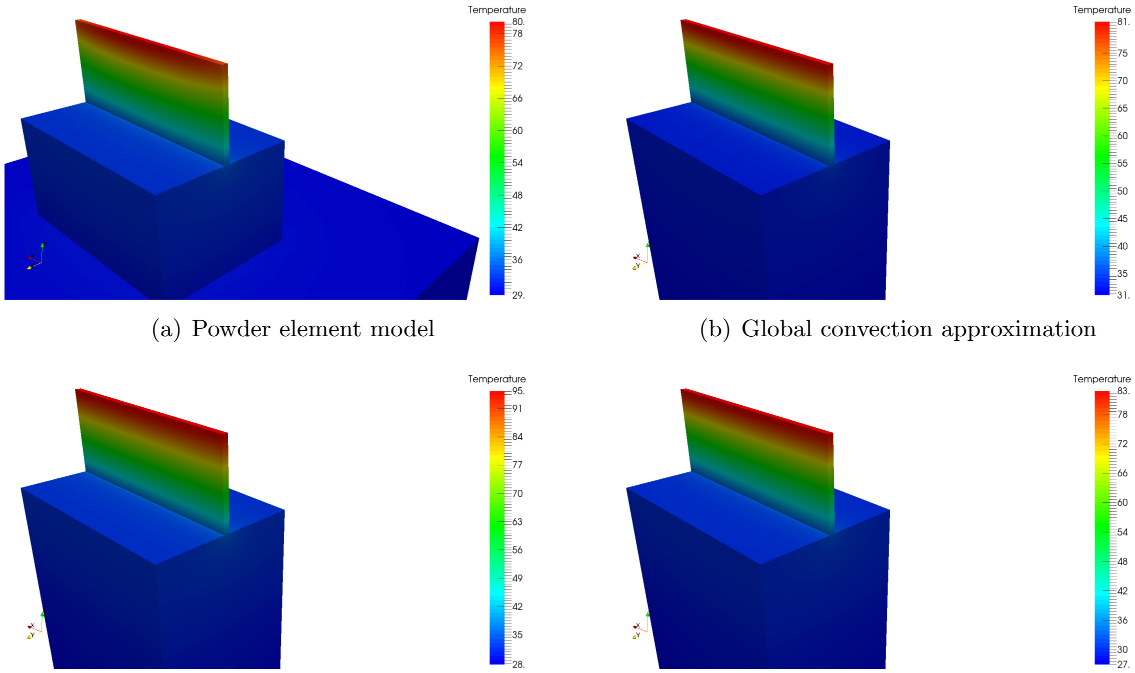
(b) Global convection approximation

(c) Regional convection approximation

(d) Regional convection and by thickness approximation

Figure 17.1: Thermal results at increment 20

At this increment, the global heat flux approximation does not agree with the powder element analysis. However using the regional or the region plus by thickness convection approximations, the temperatures and gradients are nearly identical to the powder case. Now look at the thermal history at the end of the simulation, shown in Figure 17.2.



(a) Powder element model

(b) Global convection approximation

(c) Regional convection approximation

(d) Regional convection and by thickness approximation

Figure 17.2: Thermal results at final increment

At this time step the global convection value does match with the powder case very well. However the regional case, while very accurate for the thick section, allows the top section of the part to get hotter than the powder case predicts. Using the thickness based convection corrects for this, bringing the temperature very close to the powder case for both the thick and then sections, while taking roughly 1/3 as long to complete.

Example 18

Automatic Homogenization of STLs

All of the files required to execute this example are in the `Local Simulation Example Files.zip` which can be downloaded from the Tutorials Download Page.

18.1 Problem Description

Figures 18.1 shows the geometry used in the present STL homogenization example. The component is a vertical cylinder with 3 spokes jutting from the sides, cut at a 45 degree angle. Each of the 3 spokes have support structures. The 1st set of supports is a solid support with a 0.100 mm wall thickness. The 2nd support structure is a loosely meshed zero thickness STL. The 3rd support structure is a finely meshed zero thickness STL. In the center of the cylinder a latticed structure component is also built. All 3 supports and the latticed structure will be modeled using automatic homogenization via the *STLH card. The present simulation uses Inconel 625 material properties for the build and the build plate. Support structure failure is also considered at an arbitrary support structure failure of 1800 MPa, assigned by the *UTSR card.

STLH takes any arbitrary STL file and homogenizes the part, creating volumetric representation of the part's bounding box. To account for the differences in thermal and mechanical behavior between the shrink-wrapped volume and the original geometry, the material properties are scaled by the volume fraction. The volume fraction is simply the ratio of the original as printed volume to the homogenized volume. There are 4 options to assign the volume fraction:

- User specified volume fraction
- For closed, volumetric type supports and lattices, the volume of the component can be calculated directly from the STL file geometry
- Calculate the volume of the component by specifying a structure wall thickness
- Calculate the volume of the component based upon the laser beam diameter used in the source PRM file

This example uses all 4 options:

- Lattice - Calculate volume from STL directly
- Solid support - User specified volume fraction, set to 0.22 by the *STLM card
- Loose zero thickness support - Calculate volume fraction from a specified wall thickness, set to 0.22 mm
- Fine zero thickness support - Calculate volume fraction from laser beam diameter, which is 0.15 mm for this PRM file.

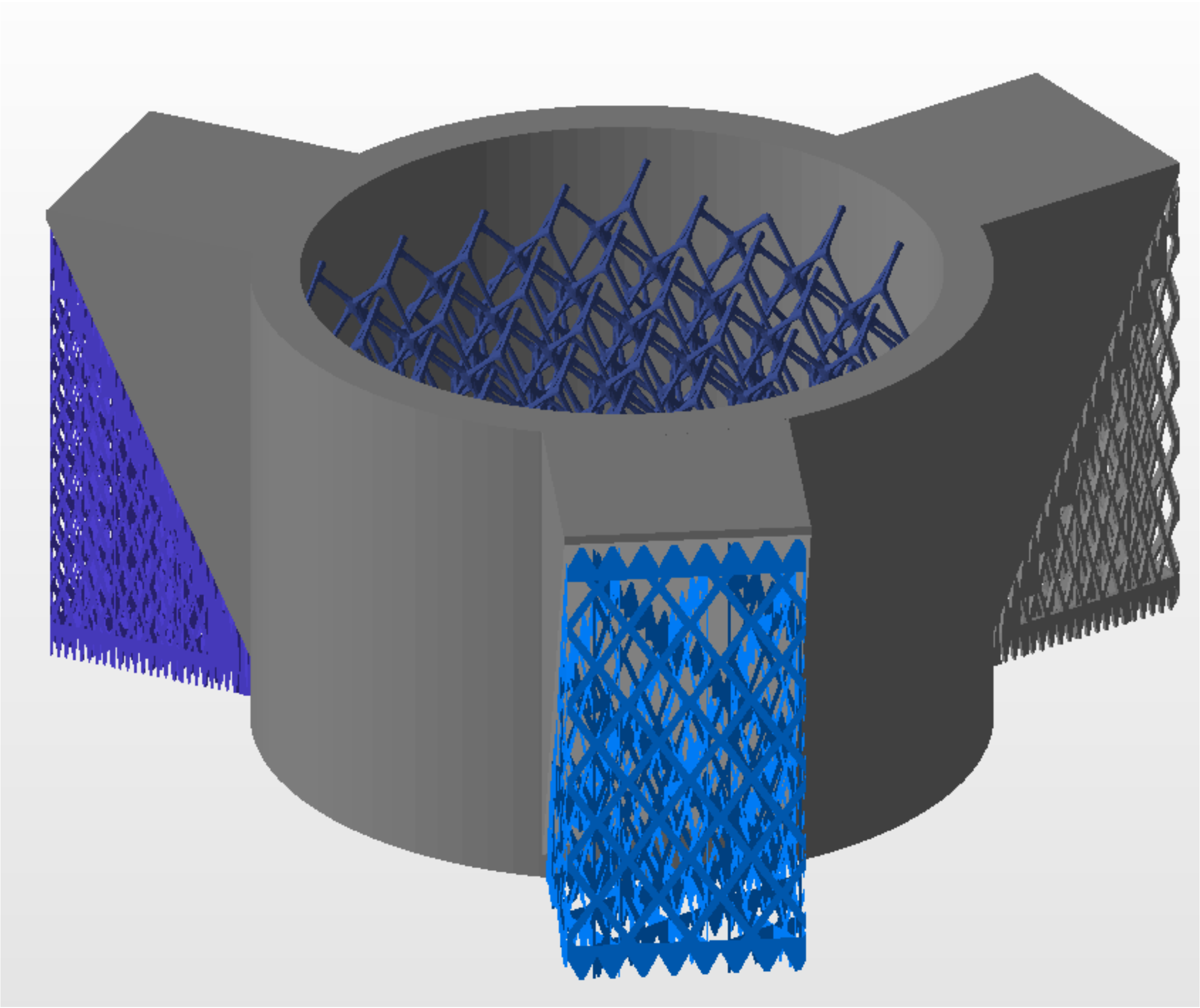


Figure 18.1: STL homogenization example geometry

The *STLH card is used to map the volume fraction choices to the STL files. It has additional control, the alpha radius, which specifies a spherical radius, which sets the maximum hole size that will be homogenized. For instance the fine zero thickness supports have an alpha radius of 5 mm, so that if any gap exists which can fit a sphere with a 5 mm diameter across it, that gap will not be filled in during meshing.

Figure 18.2 shows the mesh and the structure type of the part. Observe the homogenization of the lattice and support structures.

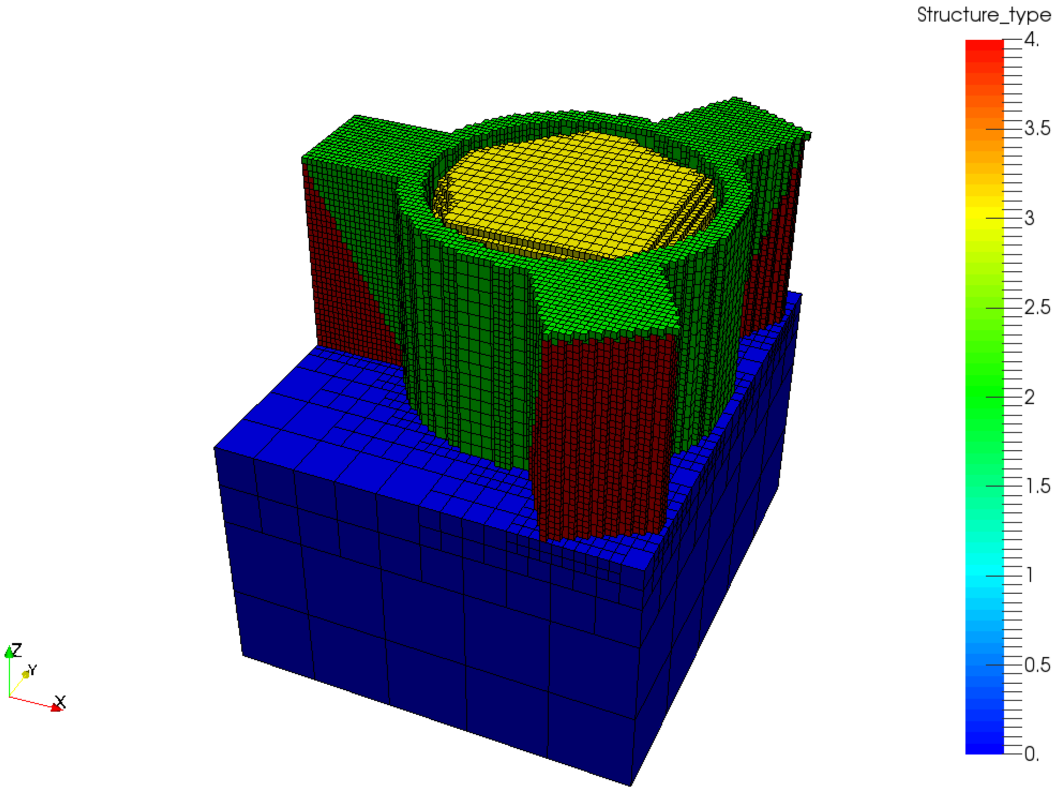


Figure 18.2: STL homogenized mesh with structure type

These values correspond as follows:
0 - Substrate or build plate
1 - Powder
2 - Component
3 - Homogenized component 4 - Support structure
5 - Failed support structure

18.2 Running Netfabb Simulation

18.2.1 Thermal Analysis

To run the model, from a command line run:

```
$ pan -b STLH_t
```

The analysis progress is written on file `STLH_t.out`. To check progress run:

```
$ tail STLH_t.out
```

After the analysis completes, the last few lines of the output file `STLH_t.out` should be similar to the following:

```
Increment end
CPU wall for increment 33 = 00:00:01.57, since start = 00:00:42.76
 inc =       34 time =   12070.108     iter =   1 eps =  0.61816E+03
 inc =       34 time =   12070.108     iter =   2 eps =  0.12564E-11
Finished writing file results\ STLH_t_34.case
Writing record:            2, time:    12070.1080521538
Increment end
CPU wall for increment 34 = 00:00:00.64, since start = 00:00:43.40
Layer end

Mesh preview volume  =    17526.7840550381
Activated volume     =    17311.9979922638
Activated percentage =    98.7745266781411

Finished writing file .\ STLH_t.case

Analysis completed

****************************
   3 Warnings
****************************

CPU wall for printing = 00:00:23.39
CPU wall  = 00:00:43.46
CPU total = 00:01:46.79


Peak RAM used for this process = 232956 kB


END Autodesk Netfabb Local Simulation
```

Actual CPU times will differ.

18.2.2 Quasi-Static Mechanical Analysis

Run the mechanical analysis from the command line:

```
$ pan -b STLH_m
```

After the analysis completes, the last few lines of the output file `STLH_m.out` should be similar to the following:

```
------------------------------------------
 Support structure removal time increment
------------------------------------------
CPU wall for substrate removal = 00:00:02.08
 inc =       37 time =   161987.20     iter =   1 eps =  0.38096E+03
 inc =       37 time =   161987.20     iter =   2 eps =  0.19971E-09
Finished writing file results\ STLH_m_37_f.case
Finished writing file results\ STLH_m_37.case
-------------------------------

*** WARNING: 39
Support structure failure
-------------------------------
Increment end
CPU wall for increment 37 = 00:00:02.08, since start = 00:01:13.76
Layer end

------------------------------------------------------
 Total number of equilibrium iterations:           74

Mesh preview volume  =    17179.7821735255
Activated volume     =    16967.5818177669
Activated percentage =    98.7648251088680

Finished writing file .\ STLH_m_f.case
Finished writing file .\ STLH_m.case

Analysis completed

****************************
   39 Warnings
****************************

CPU wall for support removal = 00:00:02.13
CPU wall  = 00:01:13.81
CPU total = 00:04:01.61


Peak RAM used for this process = 966768 kB


END Autodesk Netfabb Local Simulation
```

Actual CPU times will differ.

Each of the warnings note a support structure element failure.

18.3 Results

Returning the log files, make note of the volume fraction assigned and calculated during the homogenization and meshing process:

```
STL file start pre-processing

Homogenizing STL 2...
Reading Lattice.stl
Reading in native format...
Binary STL file
Bounding box:
 2.172730E+00 <= x <=   2.777782E+01
 2.172791E+00 <= y <=   2.777775E+01
 5.929890E-01 <= z <=   1.943214E+01

Number of vertices  =       222600
Number of triangles =        74200
Finished reading Lattice.stl

Equivalencing vertices
Number of unique vertices =        35972
Finished vertex equivalencing

Original STL volume       =    149.531685569228


Getting Delaunay triangulation...

Number of tetrahedrons = 248850
Wall time for tetrahedralization = 0.755317


Alpha radius =    10.0000000000000
Filtering 2187 tetrahedrons...
Number of hull triangles = 2662
Finished writing binary STL file Lattice_concavity.stl
Homogenized STL volume =    8639.01118122283
Volume fraction =   1.7308889E-02

Homogenizing STL 3...
Reading Support1_Solid.stl
Reading in native format...
Binary STL file
Bounding box:
-9.002001E+00 <= x <=   1.000000E+00
 1.000000E+01 <= y <=   2.000000E+01
 0.000000E+00 <= z <=   1.943000E+01
```

```
Number of vertices  =       93636
Number of triangles =       31212
Finished reading Support1_Solid.stl

Equivalencing vertices
Number of unique vertices =       14276
Finished vertex equivalencing



Getting Delaunay triangulation...

Number of tetrahedrons = 84630
Wall time for tetrahedralization = 0.203907


Alpha radius =    15.0000000000000
Filtering 605 tetrahedrons...
Number of hull triangles = 4648
Finished writing binary STL file Support1_Solid_concavity.stl
Volume fraction =   0.2200000

Homogenizing STL 4...
Reading Support2_0Thickness_Loose.stl
Reading in native format...
Binary STL file
Bounding box:
 1.743000E+01 <= x <=   3.065200E+01
 2.484200E+01 <= y <=   3.813200E+01
 0.000000E+00 <= z <=   1.940400E+01

Number of vertices  =       17937
Number of triangles =        5979
Finished reading Support2_0Thickness_Loose.stl

Equivalencing vertices
Number of unique vertices =        5051
Finished vertex equivalencing

Calculating surface normals
Original STL surface area =    462.472214730474
Original STL volume       =    101.743887240704


Getting Delaunay triangulation...
```

```
Number of tetrahedrons = 27596
Wall time for tetrahedralization = 0.0905077


Alpha radius =    5.00000000000000
Filtering 425 tetrahedrons...
Number of hull triangles = 2928
Finished writing binary STL file Support2_0Thickness_Loose_concavity.stl
Homogenized STL volume =    924.165379014976
Volume fraction =   0.1100927

Homogenizing STL 5...
Reading Support3_0Thickness_Fine.stl
Detected repaired STL.
Reading Support3_0Thickness_Fine_ls_repair.stl instead...
Reading in native format...
Binary STL file
Bounding box:
 1.804400E+01 <= x <=   3.065600E+01
-7.656000E+00 <= y <=   4.920000E+00
-1.500000E-03 <= z <=   1.937400E+01

Number of vertices  =        90312
Number of triangles =        30104
Finished reading Support3_0Thickness_Fine_ls_repair.stl

Equivalencing vertices
Number of unique vertices =        14990
Finished vertex equivalencing

Calculating surface normals
Original STL surface area =    2520.18044128674
Original STL volume       =    378.027066193010


Getting Delaunay triangulation...

Number of tetrahedrons = 87305
Wall time for tetrahedralization = 0.25891


Alpha radius =    5.00000000000000
Filtering 1757 tetrahedrons...
Number of hull triangles = 6614
Finished writing binary STL file Support3_0Thickness_Fine_concavity.stl
Homogenized STL volume =    917.613110455625
Volume fraction =   0.4119678
```

For each of the 4 homogenized geometries the solve calculates the original STL volume and the homogenized volume. For all but the solid support structure, the volume fraction is then calculated. For the solid support structure the volume fraction has been directly assigned and is reported as 0.22. The volume fractions are:

- Lattice - 0.0173
- Solid support - 0.22
- Loose zero thickness support - 0.11
- Fine zero thickness support - 0.41

Figure 18.3 displays the displacement results of the thermo-mechanical simulation at the end of the build process, after part cool down. The part has been warped by displacement with no additional magnification. The support structure to the fore of the picture is the Solid support, assigned a volume fraction of 0.22, to the left is the Loose zero thickness support, with a calculated volume fraction of 0.11, and to the right the fine zero thickness support, with the largest volume fraction, calculated to be 0.41. The Solid and Loose supports exhibit an equivalent trend and value of distortion, while the high volume fraction support, the zero thickness Fine case, shows roughly 50% of the distortion as the less dense supports.

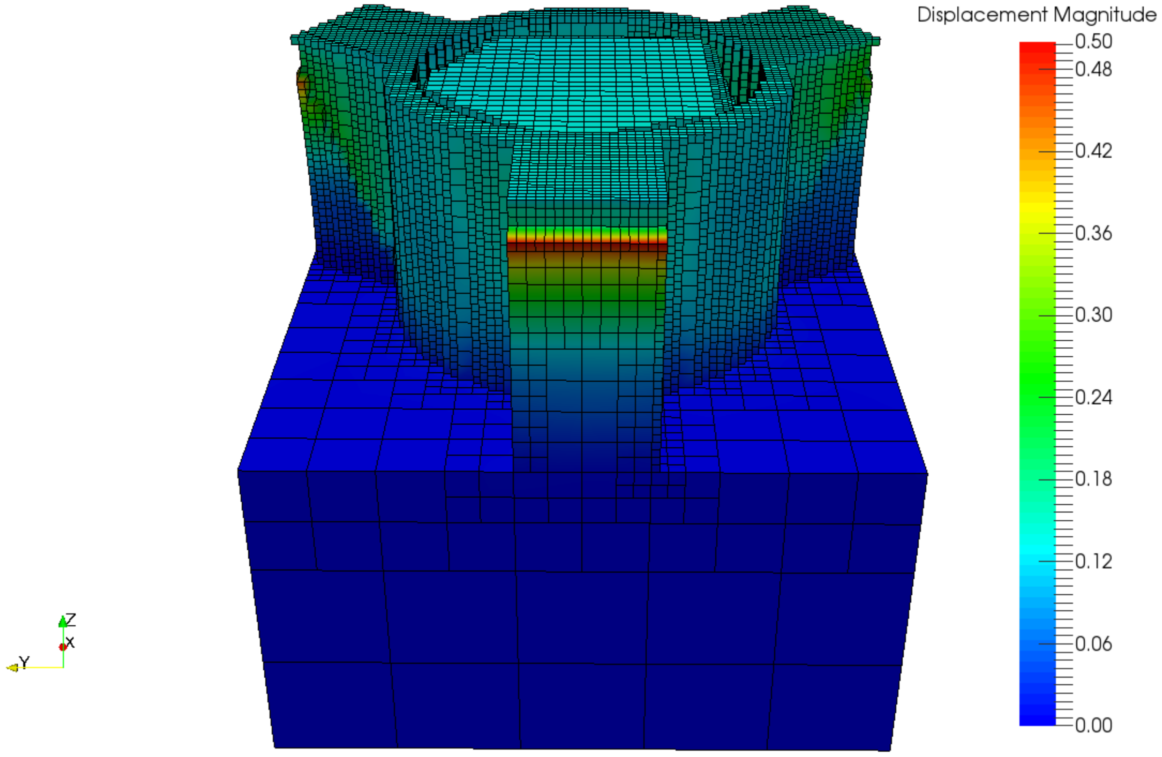


Figure 18.3: STL homogenization example displacement results

Figure 18.4 gives the structure type results at the end of the simulation, which is most useful in this case for investigating support structure failures. The figure has been filtered to only show the support structure and failed support structure types. Again the Solid and Loose support structures show a similar trend, with significant support failure along the part-support interface. The denser Fine support shows much less support failure. This illustrates the effect of different support structure volume fractions, and the usefulness of the automatic homogenization functionality.

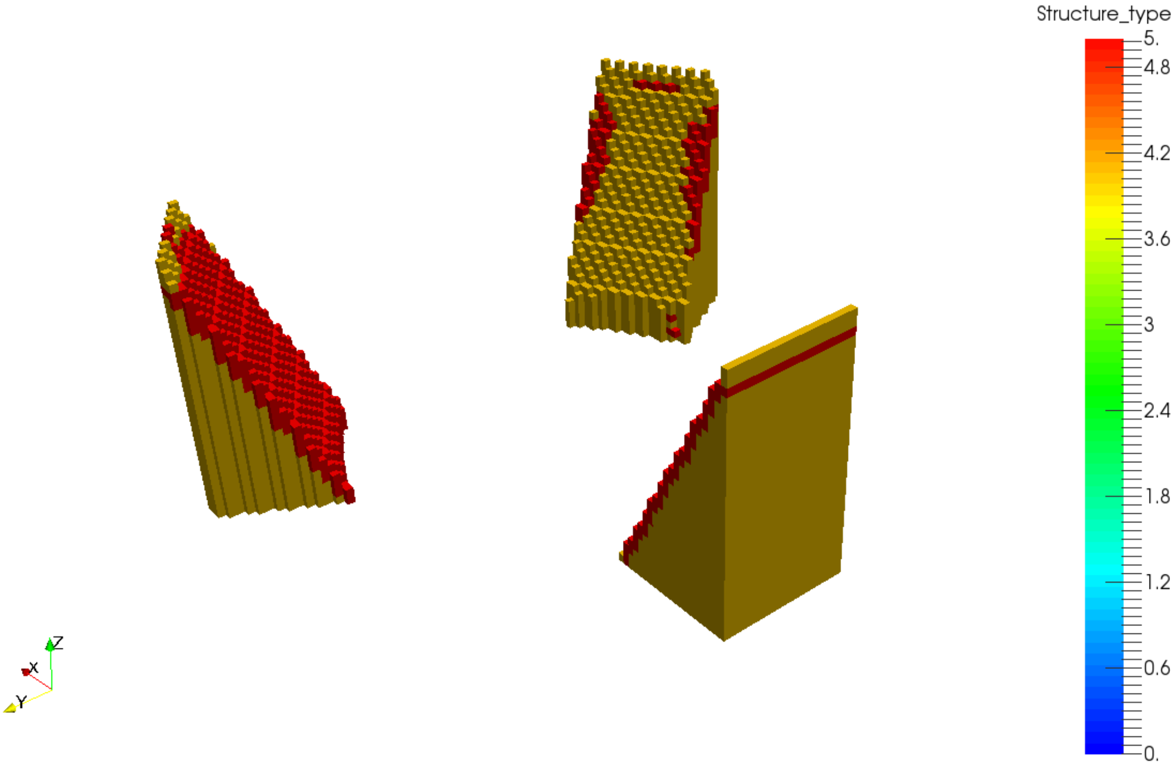


Figure 18.4: STL homogenization example support failure results