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

1 Thermo-Mechanical Process Parameter File Generation ........................................... 7
   1.1 Problem Description .......................................................... 7
   1.2 Running Netfabb Simulation .................................................. 9
   1.3 Results ................................................................. 9
   1.4 Post processing .......................................................... 10

2 Part Scale Modeling .............................................................................. 11
   2.1 Problem Description .......................................................... 11
   2.2 Running Netfabb Simulation .................................................. 13
      2.2.1 Thermal Analysis ......................................................... 13
      2.2.2 Quasi-Static Mechanical Analysis .................................. 14
   2.3 Results ................................................................. 15
   2.4 Producing distorted and compensated STL files from the simulation results ....... 15

3 Advanced Part Scale Modeling ................................................................. 19
   3.1 Problem Description .......................................................... 19
   3.2 Running Netfabb Simulation .................................................. 19
      3.2.1 Thermal Analysis ......................................................... 19
      3.2.2 Quasi-Static Mechanical Analysis .................................. 21
   3.3 Results ................................................................. 23

4 Moving adaptive refinement ........................................................................ 25
   4.1 Problem Description .......................................................... 25
   4.2 Running Netfabb Simulation .................................................. 25
      4.2.1 Thermal Analysis ......................................................... 25
   4.3 Results ................................................................. 26

5 Directed Energy Deposition ....................................................................... 27
   5.1 Problem Description .......................................................... 27
   5.2 Running Netfabb Simulation .................................................. 27
      5.2.1 Thermal Analysis ......................................................... 27
      5.2.2 Mechanical Analysis .................................................... 28
   5.3 Results ................................................................. 31

6 Part Scale Modeling with Buildplate Release ............................................... 34
   6.1 Problem Description .......................................................... 34
   6.2 Running Netfabb Simulation .................................................. 35
      6.2.1 Thermal Analysis ......................................................... 35
20.2 Running Netfabb Simulation .................................................. 131
  20.2.1 Thermal Analysis ......................................................... 131
  20.2.2 Mechanical Analysis .................................................... 132
20.3 Results ................................................................. 136
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
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.

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 filled 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
EXAMPLE 1. THERMO-MECHANICAL PROCESS PARAMETER FILE GENERATION

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:

$ prmgen 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/((mm²)°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
CPU wall for increment 34 = 00:00:00.43, since start = 00:00:14.12
  inc = 35 time = 4249.1602 iter = 1 eps = 0.23990E+03
  inc = 35 time = 4249.1602 iter = 2 eps = 0.41748E-12
Finished writing file results\02_thermal.35.case
Writing record:  2, time:  4249.16015625000
Increment end
CPU wall for increment 35 = 00:00:00.25, since start = 00:00:14.37
Layer end

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

Finished writing file .\02_thermal.case

Analysis completed

CPU wall for printing = 00:00:06.31
CPU wall = 00:00:14.43
CPU total = 00:00:29.01

Peak RAM used for this process = 90,716 kB

END Autodesk Netfabb Local Simulation

Actual CPU times will differ from system to system.
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
----------------------------------
ingc = 36 time = 6249.1602 iter = 1 eps = 0.53336E+04
inc = 36 time = 6249.1602 iter = 2 eps = 0.10599E-08

Optimizing rigid body motion...
Initial RMS displacement = 3.346287E-01
Optimized RMS displacement = 3.179544E-01
Number of optimization iterations = 250
Rotation matrix =
```
1.000000E+000 -9.670484E-006 -1.151088E-007
9.670483E-006 1.000000E+000 -2.613098E-006
1.151341E-007 2.613097E-006 1.000000E+000
```
Translation = -2.369873E-005 1.993509E-004 1.043245E-001

Finished writing file results\02_mechanical_36_f.case
Finished writing file results\02_mechanical_36.case
Increment end
CPU wall for increment 36 = 00:00:00.81, since start = 00:00:23.51
Layer end

Total number of equilibrium iterations: 72
```
Mesh preview volume = 761.062500000000
Activated volume = 761.062500000000
Activated percentage = 100.000000000000
```

Analysis completed
CPU wall for substrate removal = 00:00:00.86
CPU wall = 00:00:23.57
CPU total = 00:01:03.33

Peak RAM used for this process = 337,664 kB

END Autodesk Netfabb Local Simulation

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.

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.

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 2.2: Final distortion.
Figure 2.3: Warped STL
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.

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/(mm²°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
Figure 3.1: The Finite Element Mesh
or in Windows systems:

```bash
$ 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:

Increment end

```plaintext
CPU wall for increment 46 = 00:00:01.68, since start = 00:00:36.78
inc = 47 time = 14725.277 iter = 1 eps = 0.35216E+03
inc = 47 time = 14725.277 iter = 2 eps = 0.64861E-12
```

Finished writing file results\03_thermal.47.case

Writing record: 2, time: 14725.2769198732

Increment end

```plaintext
CPU wall for increment 47 = 00:00:00.58, since start = 00:00:37.37
Layer end
```

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

Finished writing file .\03_thermal.case

Analysis completed

```plaintext
CPU wall for printing = 00:00:24.45
CPU wall = 00:00:37.42
CPU total = 00:01:44.08
```

Peak RAM used for this process = 188,296 kB

END Autodesk Netfabb Local Simulation

Actual CPU times will differ.

### 3.2.2 Quasi-Static Mechanical Analysis

Run the analysis from the command line:

```bash
$ 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:

```
Substrate removal time increment
```

```plaintext
inc = 49 time = 24725.277 iter = 1 eps = 0.49409E+04
inc = 49 time = 24725.277 iter = 2 eps = 0.51792E-09
```
Optimizing rigid body motion...
Initial RMS displacement = 2.230159E-01
Optimized RMS displacement = 1.413638E-01
Number of optimization iterations = 241
Rotation matrix =
\[
\begin{bmatrix}
1.000000E+000 & 4.614382E-005 & 1.689121E-004 \\
-4.624778E-005 & 9.999998E-001 & 6.154971E-004 \\
-1.688836E-004 & -6.155049E-004 & 9.999998E-001
\end{bmatrix}
\]
Translation = -6.480539E-002 -7.594077E-002 1.205890E-001

Finished writing file results\03\_mechanical\_49\_f\_case
Finished writing file results\03\_mechanical\_49\_case
Increment end
CPU wall for increment 49 = 00:00:02.15, since start = 00:00:59.07
Layer end

Total number of equilibrium iterations: 97

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

Finished writing file \03\_mechanical\_f\_case
Finished writing file \03\_mechanical\_case

Analysis completed

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

CPU wall for substrate removal = 00:00:02.22
CPU wall = 00:00:59.14
CPU total = 00:03:12.24

Peak RAM used for this process = 748,136 kB

END Autodesk Netfabb Local Simulation

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 results

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.

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® 718. A surface convection of 10.d-6 W/((mm²)-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:

Starting auxspar
Number of no zeros nsymmetric =7172
Sparse preprocessing complete
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.

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:

Increment end
CPU wall for increment 4149 = 00:00:00.16, since start = 00:16:36.69
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
Writing record: 84, time: 500.000000000000
Figure 5.1: Illustration of the substrate, deposition, their dimensions in mm, the laser path (red arrows), and the coordinate system (not to scale).

Increment end
CPU wall for increment 4150 = 00:00:00.13, since start = 00:16:36.83
Layer end
Finished writing file .\05_thermal.case

Analysis completed

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

CPU wall  = 00:16:37.23
CPU total = 01:05:42.89

Peak RAM used for this process = 72,468 kB

END Autodesk Netfabb Local Simulation

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:

CPU wall for increment 4284 = 00:00:00.27, since start = 00:34:13.42
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 = 0.49331E-09
Finished writing file results\05_mechanical_4285.case
Increment end

CPU wall for increment 4285 = 00:00:00.36, since start = 00:34:13.78
Layer end

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

Analysis completed

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

CPU wall = 00:34:14.20
CPU total = 02:08:13.17

Peak RAM used for this process = 129,260 kB

END Autodesk Netfabb Local Simulation
Figure 5.2: LENS part discretized by 8-node linear hexahedral elements. Dummy boundary conditions, materials, and properties must also be applied.
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.
EXAMPLE 5. DIRECTED ENERGY DEPOSITION

Figure 5.3: Temperature results ($^\circ$C) at two sample increments.

(a) Increment 1000

(b) Increment 4000
(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® 718 is 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.](image)

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/(mm²)°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:

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

If the system is linux, from the terminal run:

```bash
$ 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:

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

or in Windows systems:

```bash
$ 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
CPU wall for increment 14 = 00:00:02.18, since start = 00:01:08.40
  inc = 15 time = 57672.801 iter = 1 eps = 0.10940E+02
  inc = 15 time = 57672.801 iter = 2 eps = 0.13226E-11
Finished writing file results\06_thermal.15.case
Writing record: 3, time: 57672.8014167935
Increment end
CPU wall for increment 15 = 00:00:02.19, since start = 00:01:10.60
Layer end

Mesh preview volume = 190987.524817100
Activated volume = 190987.524817100
Activated percentage = 100.0000000000000

Finished writing file .\06_thermal.case

Analysis completed
```
**EXAMPLE 6. PART SCALE MODELING WITH BUILDPLATE RELEASE**

1 Warning

CPU wall for printing = 00:00:35.28
CPU wall = 00:01:10.67
CPU total = 00:02:37.40

Peak RAM used for this process = 603,252 kB

END Autodesk Netfabb Local Simulation

### 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 = 17 time = 157672.80 iter = 1 eps = 0.21516E+06
   inc = 17 time = 157672.80 iter = 2 eps = 0.15258E-07

Optimizing rigid body motion...
Initial RMS displacement = 1.873007E+00
Optimized RMS displacement = 1.170165E+00
Number of optimization iterations = 278
Rotation matrix =
   9.999818E-001  2.828207E-004  -6.022653E-003
   -2.776064E-004  9.999996E-001  8.665997E-004
   6.022895E-003  -8.649120E-004  9.999815E-001
Translation =
   1.182008E-001  -6.113179E-002  -6.271939E-001

Finished writing file results\06_mechanical_17.f.case
Finished writing file results\06_mechanical_17_case
Increment end
CPU wall for increment 17 = 00:00:10.43, since start = 00:02:44.03
Layer end

----------------------------------
Total number of equilibrium iterations: 29
```

Mesh preview volume = 190987.524817100
Activated volume = 190987.524817100
Activated percentage = 100.000000000000
Finished writing file .\06.mechanical_f.case
Finished writing file .\06.mechanical.case

Analysis completed

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

CPU wall for substrate removal = 00:00:10.48
CPU wall = 00:02:44.09
CPU total = 00:08:46.61

Peak RAM used for this process = 3,202,404 kB

END Autodesk Netfabb Local Simulation

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.
EXAMPLE 6. PART SCALE MODELING WITH BUILDPLATE RELEASE

(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.

![Bracket with and without supports](image)

(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 $25.\text{d-6 W/}(\text{mm}^2\text{°C})$ 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:

```
inc = 29 time = 110026.96 iter = 1 eps = 0.12597E+04
ing = 29 time = 110026.96 iter = 2 eps = 0.11532E-11
Finished writing file results\07.thermal.29.case
Writing record: 2, time: 110026.958120637
Increment end
CPU wall for increment 29 = 00:00:00.73, since start = 00:00:39.46
Layer end
Mesh preview volume = 110852.789200311
Activated volume = 110852.789200311
Activated percentage = 100.000000000000
Finished writing file .\07.thermal.case
```

Analysis completed

```
1 Warning
```

```
1 Critical warning
```

CPU wall for printing = 00:00:21.41
CPU wall = 00:00:39.51
CPU total = 00:01:37.65

Peak RAM used for this process = 330,204 kB

END Autodesk Netfabb Local Simulation

Actual CPU times will differ. The Warning indicates the simulation time needed to be adjusted. The Critical Warning informs users that the CLI type support structures have been deprecated, and that STL type supports are suggested.

### 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:

*The warnings here are the same as given in the thermal log file.*

Actual CPU times will differ. The warnings here are the same as given in the thermal log file.

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:

```
------------------------------------------
Support structure removal time increment
------------------------------------------
inc = 32 time = 260026.96 iter = 1 eps = 0.29774E+04
inc = 32 time = 260026.96 iter = 2 eps = 0.23414E-08
```

Optimizing rigid body motion...

```
Initial RMS displacement = 4.536035E-01
Optimized RMS displacement = 3.984455E-01
Number of optimization iterations = 255
Rotation matrix =
9.999940E-01  -7.913967E-05  3.466786E-03
7.063056E-05   9.99970E-01  2.454520E-03
-3.466970E-03  -2.454260E-03  9.999910E-01
Translation =    -8.765369E-02   -4.793811E-02   -4.500820E-01
```

Finished writing file results\07_mechanical2_32_f.case
Finished writing file results\07_mechanical2_32.case
Increment end
CPU wall for increment 32 = 00:00:02.86, since start = 00:01:21.50
Layer end

------------------------------------------------------------------------
Total number of equilibrium iterations: 63

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

Signal tag 6DB7
*** CRITICAL WARNING: 2
Recoater Interference Detected at 3 layer groups. Minimum clearance of -22.5280761718771 at height 24.0000000000000 mm.

Finished writing file .\07_mechanical2.f.case
Finished writing file .\07_mechanical2.case

Analysis completed

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

*******************************
2 Critical warnings
*******************************

CPU wall for support removal = 00:00:02.91
CPU wall = 00:01:21.56
CPU total = 00:04:31.89

Peak RAM used for this process = 1,273,032 kB
END Autodesk Netfabb Local Simulation

The warnings from the previous simulations are shown here as well, in addition to a Recoater Interference Critical Warning, along with several Recoater Interference Warnings and numerous Support structure failure warnings.

7.3 Results

There are an additional log file created during simulation file name recoater.txt. Below are the results seen in the 07_mechanical1_recoater.txt:
### EXAMPLE 7. PART SCALE MODELING WITH CLI SUPPORT STRUCTURES

####time (s), layer group, recoater clearance (%), top z deformed coord (mm), recoater coord (mm), top z undeformed coord (mm)

<table>
<thead>
<tr>
<th>Time (s)</th>
<th>Layer Group</th>
<th>Clearance (%)</th>
<th>Top Z Deformed Coord (mm)</th>
<th>Recoater Coord (mm)</th>
<th>Top Z Undeformed Coord (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.172554E+04</td>
<td>1</td>
<td>77.742</td>
<td>4.008903E+00</td>
<td>4.040000E+00</td>
<td>4.000000E+00</td>
</tr>
<tr>
<td>2.389416E+04</td>
<td>2</td>
<td>58.549</td>
<td>8.016581E+00</td>
<td>8.040000E+00</td>
<td>8.000000E+00</td>
</tr>
<tr>
<td>3.607386E+04</td>
<td>3</td>
<td>47.107</td>
<td>1.202116E+01</td>
<td>1.204000E+01</td>
<td>1.200000E+01</td>
</tr>
<tr>
<td>4.815440E+04</td>
<td>4</td>
<td>70.059</td>
<td>1.601198E+01</td>
<td>1.604000E+01</td>
<td>1.600000E+01</td>
</tr>
<tr>
<td>6.195965E+04</td>
<td>5</td>
<td>64.166</td>
<td>2.001433E+01</td>
<td>2.004000E+01</td>
<td>2.000000E+01</td>
</tr>
<tr>
<td>7.656102E+04</td>
<td>6</td>
<td>59.579</td>
<td>2.401617E+01</td>
<td>2.404000E+01</td>
<td>2.400000E+01</td>
</tr>
<tr>
<td>9.112844E+04</td>
<td>7</td>
<td>59.302</td>
<td>2.801628E+01</td>
<td>2.804000E+01</td>
<td>2.800000E+01</td>
</tr>
<tr>
<td>1.035748E+05</td>
<td>8</td>
<td>78.580</td>
<td>3.200857E+01</td>
<td>3.204000E+01</td>
<td>3.200000E+01</td>
</tr>
<tr>
<td>1.092742E+05</td>
<td>9</td>
<td>79.286</td>
<td>3.600829E+01</td>
<td>3.604000E+01</td>
<td>3.600000E+01</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>Time (s)</th>
<th>Layer Group</th>
<th>Clearance (%)</th>
<th>Top Z Deformed Coord (mm)</th>
<th>Recoater Coord (mm)</th>
<th>Top Z Undeformed Coord (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.172554E+04</td>
<td>1</td>
<td>77.742</td>
<td>4.008903E+00</td>
<td>4.040000E+00</td>
<td>4.000000E+00</td>
</tr>
<tr>
<td>2.389416E+04</td>
<td>2</td>
<td>58.549</td>
<td>8.016581E+00</td>
<td>8.040000E+00</td>
<td>8.000000E+00</td>
</tr>
<tr>
<td>3.607386E+04</td>
<td>3</td>
<td>47.107</td>
<td>1.202098E+01</td>
<td>1.204000E+01</td>
<td>1.200000E+01</td>
</tr>
<tr>
<td>4.815440E+04</td>
<td>4</td>
<td>69.893</td>
<td>1.601204E+01</td>
<td>1.604000E+01</td>
<td>1.600000E+01</td>
</tr>
<tr>
<td>6.195965E+04</td>
<td>5</td>
<td>0.675</td>
<td>2.003973E+01</td>
<td>2.004000E+01</td>
<td>2.000000E+01</td>
</tr>
<tr>
<td>7.656102E+04</td>
<td>6</td>
<td>-22.528</td>
<td>2.404901E+01</td>
<td>2.404000E+01</td>
<td>2.400000E+01</td>
</tr>
<tr>
<td>9.112844E+04</td>
<td>7</td>
<td>11.342</td>
<td>2.803546E+01</td>
<td>2.804000E+01</td>
<td>2.800000E+01</td>
</tr>
<tr>
<td>1.035748E+05</td>
<td>8</td>
<td>77.417</td>
<td>3.200903E+01</td>
<td>3.204000E+01</td>
<td>3.200000E+01</td>
</tr>
<tr>
<td>1.092742E+05</td>
<td>9</td>
<td>78.456</td>
<td>3.600862E+01</td>
<td>3.604000E+01</td>
<td>3.600000E+01</td>
</tr>
</tbody>
</table>

Note that when support structure failure is taken into account, the likelihood of a catastrophic recoater interference event becomes very high.

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.

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 the simulation log. 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:
EXAMPLE 8. POWDER BED MOVING SOURCE MODELING WITH CUSTOM TOOLTIPSHS

$ type t.out

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

CPU wall for increment 246 = 00:00:00.13, since start = 00:05:32.55
inc = 247 time = 60.000000 iter = 1 eps = 0.46496E-03
Finished writing file results\ t_247.case
Increment end
CPU wall for increment 247 = 00:00:00.14, since start = 00:05:32.69
Finished writing file \ t.case

Analysis completed

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

CPU wall = 00:05:33.04
CPU total = 00:18:16.82

Peak RAM used for this process = 536,444 kB

END Autodesk Netfabb Local Simulation

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 =136,319
Sparse preprocessing complete
inc = 244 time = 60.000000 iter = 1 eps = 0.37673E-03
Finished writing file results\ tadpm_244.case
Increment end
CPU wall for increment 244 = 00:00:00.11, since start = 00:03:17.61
Finished writing file \ tadpm.case

Analysis completed

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

CPU wall = 00:03:17.66
CPU total = 00:12:26.30
EXAMPLE 8. POWDER BED MOVING SOURCE MODELING WITH CUSTOM TOOLPATHS

Peak RAM used for this process = 119,360 kB

END Autodesk Netfabb Local Simulation

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
    CPU wall for increment 247 = 00:00:00.14, since start = 00:04:46.39
    Layer end
    Finished writing file .\ tdirect.case

Analysis completed

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

CPU wall = 00:04:46.44
CPU total = 00:18:21.07

Peak RAM used for this process = 137,964 kB

END Autodesk Netfabb Local Simulation

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
    CPU wall for increment 244 = 00:00:00.25, since start = 00:08:49.24
    Layer end
    Finished writing file .\ tddmp.case

Analysis completed

***************************************************************************
  1 Warning
***************************************************************************
CPU wall  = 00:08:49.60  
CPU total = 00:34:24.76

Peak RAM used for this process = 196,736 kB

END Autodesk Netfabb Local Simulation

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.

![Substrate Heating Simulations](image)

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 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).
Figure 8.2: Thermal simulation of powder sintering
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 finite element mesh](image)

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/((mm²)°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:

Increment end
CPU wall for increment 25 = 00:00:01.06, since start = 00:00:19.66
inc = 26 time = 1982.8118 iter = 1 eps = 0.14964E+03
inc = 26 time = 1982.8118 iter = 2 eps = 0.61878E-12
Finished writing file results\09_thermal_26.case
Writing record: 2, time: 1982.81176470588
Increment end
CPU wall for increment 26 = 00:00:00.42, since start = 00:00:20.08
Layer end

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

Finished writing file .\09_thermal.case

Analysis completed

CPU wall for printing = 00:00:09.53
CPU wall = 00:00:20.14
CPU total = 00:00:44.66

Peak RAM used for this process = 146,772 kB

END Autodesk Netfabb Local Simulation

Actual CPU times will differ.
9.2.2 Quasi-Static Mechanical Analysis

Run the first analysis from the command line:

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

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

```bash
$ 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 = 28  time = 101982.81  iter = 1  eps = 0.38806E+04
  inc = 28  time = 101982.81  iter = 2  eps = 0.23988E-08

Optimizing rigid body motion...
Initial RMS displacement = 5.19442E-01
Optimized RMS displacement = 5.14089E-01
Number of optimization iterations = 207
Rotation matrix =
  1.000000E+000  6.764603E-006  6.375272E-007
  -6.764607E-006  1.000000E+000  7.124815E-006
  -6.374790E-007  -7.124820E-006  1.000000E+000
Translation =
  1.117124E-004  -1.287760E-004  7.433468E-002
Finished writing file results\09_mechanical_28_f.case
Finished writing file results\09_mechanical_28_case
Increment end
CPU wall for increment 28 = 00:00:01.26, since start = 00:00:30.80
Layer end

----------------------------------
Total number of equilibrium iterations: 56
----------------------------------
Mesh preview volume = 791.560000000000
Activated volume = 791.560000000000
Activated percentage = 100.000000000000

Signal tag 3743
*** CRITICAL WARNING: 2
  Code 1041
Recoater interference detected at four layer groups. Minimum clearance of 52.767 percent at height 5.600 mm.

Finished writing file .\09_mechanical_f.case
Finished writing file .\09_mechanical_case
Analysis completed

******************************
5 Warnings
******************************

******************************
2 Critical warnings
******************************

CPU wall for substrate removal = 00:00:01.31
CPU wall = 00:00:30.85
CPU total = 00:01:26.81

Peak RAM used for this process = 595,676 kB

END Autodesk Netfabb Local Simulation

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:

----------------------
*COOL time increment
----------------------

CPU wall for printing = 00:00:17.70
HTOR is being set to zero***
inc = 27 time = 51982.812 iter = 1 eps = 0.23906E+05
inc = 27 time = 51982.812 iter = 2 eps = 0.88357E-09

Finished writing file results\09_mechanical_ppla_27.case
Increment end
CPU wall for increment 27 = 00:00:01.20, since start = 00:00:31.04
Layer end
CPU wall for cooldown = 00:00:01.20

--------------------------------------
Plasticity iteration # 1
EXAMPLE 9. POWDER BED PART LEVEL PLASTICITY

Signal tag 5768
*** CRITICAL WARNING: 2
Code 1028
Residual is increasing. Reducing time step.

Switching plasticity algorithm

------------------------
Plasticity progress 0.333333333333333
-------------------------------
Plasticity progress 0.666666666666667
-------------------------------
Plasticity progress 1.000000000000000
-------------------------------

Signal tag 50A1
*** CRITICAL WARNING: 3
Code 1028
Residual is increasing. Reducing time step.

Reducing plasticity step
Relaxation factor: 0.500000000000000
New plasticity step: 0.166666666666667
EXAMPLE 9. POWDER BED PART LEVEL PLASTICITY

Plasticity progress 0.833333333333333
inc = 28 time = 93649.478 iter = 1 eps = 0.10225E+04
inc = 28 time = 93649.478 iter = 2 eps = 0.52567E+03
inc = 28 time = 93649.478 iter = 3 eps = 0.31986E+02
inc = 28 time = 93649.478 iter = 4 eps = 0.49688E+00
inc = 28 time = 93649.478 iter = 5 eps = 0.44681E-02

Plasticity progress 1.000000000000000
inc = 28 time = 101982.81 iter = 1 eps = 0.35153E+04
inc = 28 time = 101982.81 iter = 2 eps = 0.11223E+04
inc = 28 time = 101982.81 iter = 3 eps = 0.45928E+03
inc = 28 time = 101982.81 iter = 4 eps = 0.35233E+02
inc = 28 time = 101982.81 iter = 5 eps = 0.60754E+01
inc = 28 time = 101982.81 iter = 6 eps = 0.31799E+00
inc = 28 time = 101982.81 iter = 7 eps = 0.21561E-02

Finished writing file results\09_mechanical_ppla_28_f.case
Finished writing file results\09_mechanical_ppla_28.case
Increment end
CPU wall for increment 28 = 00:00:22.64, since start = 00:00:53.68
Layer end
CPU wall for plasticity = 00:00:18.96

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

-------------------------------
inc = 29 time = 151982.81 iter = 1 eps = 0.19473E+04
inc = 29 time = 151982.81 iter = 2 eps = 0.18681E-08

Optimizing rigid body motion...
Initial RMS displacement = 4.204903E-01
Optimized RMS displacement = 4.141274E-01
Number of optimization iterations = 231
Rotation matrix =
1.000000E+000  1.190363E-005  4.425130E-007
-1.190363E-005  1.000000E+000  8.177796E-006
-4.424156E-007 -8.177774E-006  1.000000E+000
Translation =
1.011255E-004 -2.356972E-004  7.283104E-002

Finished writing file results\09_mechanical_ppla_29_f.case
Finished writing file results\09_mechanical_ppla_29.case
Increment end
CPU wall for increment 29 = 00:00:01.37, since start = 00:00:55.06
Layer end

Total number of equilibrium iterations: 76
EXAMPLE 9. POWDER BED PART LEVEL PLASTICITY

Mesh preview volume  =  791.560000000000
Activated volume     =  791.560000000000
Activated percentage =  100.000000000000

Signal tag 7714
*** CRITICAL WARNING: 4
Code 1041
Recoater interference detected at four layer groups. Minimum clearance of 52.767 percent at height 5.600 mm.

Finished writing file .\09_mechanical.ppla_f.case
Finished writing file .\09_mechanical.ppla.case

Analysis completed

******************************
5 Warnings
******************************

******************************
4 Critical warnings
******************************

CPU wall for substrate removal = 00:00:01.42
CPU wall     = 00:00:55.12
CPU total    = 00:02:57.52

Peak RAM used for this process = 748,208 kB

END Autodesk Netfabb Local Simulation

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.
EXAMPLE 9. POWDER BED PART LEVEL PLASTICITY

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/((mm²)°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/((mm²)°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:

```
inc = 649 time = 461.83593 iter = 1 eps = 0.67771E-04
Finished writing file results\ t10_moving_649.c.case
Increment end
CPU wall for increment 649 = 00:00:00.19, since start = 00:07:43.29

Starting refine
Number of refined nodes = 5708
Number of refined elements = 4200
Number of equations = 5446
Number of constrained eqns = 262

Starting auxspar
Number of no zeros nsymmetric =122,380
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
CPU wall for increment 650 = 00:00:00.21, since start = 00:07:43.50
Finished writing file .\ t10_moving.case
Finished writing file .\ t10_moving_c.case

Analysis completed
CPU wall = 00:07:43.85
CPU total = 00:30:09.72

Peak RAM used for this process = 129,344 kB

END Autodesk Netfabb Local Simulation

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:

```
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
CPU wall for increment 231 = 00:00:00.08, since start = 00:01:08.42
Finished writing file .\ t10_multilayer.case
Finished writing file .\ t10_multilayer_c.case
```

Analysis completed

```
CPU wall = 00:01:08.47
CPU total = 00:04:06.41
```

Peak RAM used for this process = 97,428 kB

END Autodesk Netfabb Local Simulation

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.

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.
Figure 10.1: *LFUS results moving adaptive mesh

(a) Peak temperatures

(b) Melt indicator
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 |
| 2.110243E-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 |
| 2.110243E-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 |
| 2.110243E-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 |
| 2.110243E-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 |
| 2.110243E-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 |
| 2.110243E-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 |
| 2.110243E-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 |
| 2.110243E-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 |
| 2.110243E-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 |
| 2.110243E-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 |
| 2.110243E-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 |
| 2.110243E-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 |
| 2.110243E-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 |
| 2.110243E-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 |
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.
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
# 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
2.310243E-04, 4.616060E-01, 5.228448E-02, 0.000000E+00, 1.047308E+03, 1, 0
2.310243E-04, 4.616060E-01, 5.228448E-02, 0.000000E+00, 1.047308E+03, 1, 0
2.510243E-04, 4.420075E-01, 5.627184E-02, 0.000000E+00, 1.064964E+03, 1, 0
2.510243E-04, 4.420075E-01, 5.627184E-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
3.110243E-04, 3.832120E-01, 4.430976E-02, 0.000000E+00, 1.069138E+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 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 build plate from the machine after the deposition process is complete, but before the part is removed from the build plate. The mesh, shown with and without the support elements, is shown in Figures 11.1.

![Figure 11.1: Sphere with and without supports.](image)

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:

```bash
$ pan -b t11
```

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

```bash
$ tail t11.out
```

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

```
Mesh preview volume = 4487.62499999999
Activated volume = 4487.62499999999
Activated percentage = 100.000000000000

Finished writing file .\t11.case

Analysis completed

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

CPU wall for printing = 00:00:23.22
CPU wall = 00:00:34.46
CPU total = 00:01:41.33

Peak RAM used for this process = 181,672 kB

END Autodesk Netfabb Local Simulation

END Autodesk Netfabb Local Simulation
```

Actual CPU times will differ.
11.2.2 Quasi-Static Mechanical Analysis

Run the mechanical analysis from the command line:

```bash
$ pan -b m11
```

After the analysis completes, the last few lines of the output file `multist1_mech.out` should be similar to the following:

```
Support structure removal time increment
------------------------------------------
  inc = 40  time = 18831.914  iter = 1  eps = 0.29089E+03
  inc = 40  time = 18831.914  iter = 2  eps = 0.31843E-09

Optimizing rigid body motion...
Initial RMS displacement = 1.045277E-01
Optimized RMS displacement = 1.030341E-01
Number of optimization iterations = 235
Rotation matrix =
  1.000000E+000  -5.369748E-007  7.783688E-006
  5.369652E-007  1.000000E+000  1.233429E-006
 -7.783688E-006  -1.233425E-006  1.000000E+000
Translation =
  9.521316E-005  3.760801E-006  1.760589E-002
```

Finished writing file results\ m11_40.f.case
Finished writing file results\ m11_40.case
Increment end
CPU wall for increment 40 = 00:00:01.99, since start = 00:01:10.28
Layer end

Total number of equilibrium iterations: 79

Mesh preview volume = 4487.6249999999
Activated volume = 4487.6249999999
Activated percentage = 100.000000000000

Finished writing file .\ m11.f.case
Finished writing file .\ m11.case

Analysis completed

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

**************************
1 Critical warning
**************************
**Example 11. Modeling Support Structures Using Multiple STLS**

***************

CPU wall for support removal = 00:00:02.04  
CPU wall = 00:01:10.33  
CPU total = 00:04:05.03

Peak RAM used for this process = 801,804 kB

END Autodesk Netfabb Local Simulation

Actual CPU times will differ.

Each of the warnings note a support structure element failure.

### 11.3 Results

First look at the m11_recoater.txt file to investigate possible build failure.

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

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.

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

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.
Figure 11.2: Structure type results

(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 $0.01 \times$ that of the solid while specific heat is $0.6 \times$ 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 $25.\text{d-6} \text{ W/}((\text{mm}^2)\text{°C})$ 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 $25.\text{d-6} \text{ W/}((\text{mm}^2)\text{°C})$.

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.
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:

```
CPU wall for increment 37 = 00:00:01.39, since start = 00:00:36.05
  inc =  38 time =  15831.914  iter =  1  eps =  0.45808E+02
  inc =  38 time =  15831.914  iter =  2  eps =  0.13437E-12
Finished writing file results\ t0_38.case
Writing record:  2, time:  15831.9140625000
Increment end
CPU wall for increment 38 = 00:00:00.58, since start = 00:00:36.63
Layer end

Mesh preview volume = 4487.624999999999
Activated volume = 4487.624999999999
Activated percentage = 100.000000000000

Finished writing file .\ t0.case

Analysis completed

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

CPU wall for printing = 00:00:23.64
CPU wall = 00:00:36.68
CPU total = 00:01:43.92

Peak RAM used for this process = 201,740 kB

END Autodesk Netfabb Local Simulation

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:

CPU wall for increment 37 = 00:00:02.26, since start = 00:01:07.12
inc = 38 time = 15831.914 iter = 1 eps = 0.44958E+02
inc = 38 time = 15831.914 iter = 2 eps = 0.13987E-12
Finished writing file results\ t1_38.case
Writing record: 2, time: 15831.9140625000
Increment end
CPU wall for increment 38 = 00:00:01.05, since start = 00:01:08.17
Layer end

Mesh preview volume = 4487.62499999999
Activated volume = 4487.62499999999
Activated percentage = 100.000000000000

Finished writing file \t1.case

Analysis completed

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

CPU wall for printing = 00:00:33.06
CPU wall = 00:01:08.23
CPU total = 00:02:38.15

Peak RAM used for this process = 503,436 kB

END Autodesk Netfabb Local Simulation

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:

------------------------------------------
Support structure removal time increment
------------------------------------------
inc = 40 time = 18831.914 iter = 1 eps = 0.25654E+03
Example 12. Multi-scale Powder Bed Simulations with Powder

\[ \text{inc} = 40 \quad \text{time} = 18831.914 \quad \text{iter} = 2 \quad \text{eps} = 0.37390E-09 \]

Optimizing rigid body motion...
Initial RMS displacement = 9.425852E-02
Optimized RMS displacement = 9.416243E-02
Number of optimization iterations = 254
Rotation matrix =
\[
\begin{pmatrix}
1.000000E+000 & -8.785314E-008 & 7.088365E-006 \\
8.772185E-008 & 1.000000E+000 & 1.852115E-005 \\
-7.088366E-006 & -1.852114E-005 & 1.000000E+000
\end{pmatrix}
\]
Translation = 9.830615E-005 1.326112E-004 4.245931E-003

Finished writing file results \ m0.40_f.case
Finished writing file results \ m0.40.case
Increment end
CPU wall for increment 40 = 00:00:01.94, since start = 00:01:16.89
Layer end

Total number of equilibrium iterations: 79

Mesh preview volume = 4487.6249999999
Activated volume = 4487.6249999999
Activated percentage = 100.000000000000

Finished writing file .\ m0.f.case
Finished writing file .\ m0.case

Analysis completed

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

***************
1 Critical warning
***************

CPU wall for support removal = 00:00:01.99
CPU wall = 00:01:16.94
CPU total = 00:04:20.45

Peak RAM used for this process = 843,580 kB

END Autodesk Netfabb Local Simulation
Actual CPU times will differ. Each of the warnings note a support structure element failure. The critical warning notes that 100% of the interface supports have failed.

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

<table>
<thead>
<tr>
<th>inc</th>
<th>time</th>
<th>iter</th>
<th>eps</th>
</tr>
</thead>
<tbody>
<tr>
<td>40</td>
<td>18831.914</td>
<td>1</td>
<td>0.22759E+03</td>
</tr>
<tr>
<td>40</td>
<td>18831.914</td>
<td>2</td>
<td>0.34243E-09</td>
</tr>
</tbody>
</table>

Optimizing rigid body motion...

Initial RMS displacement = 9.236756E-02
Optimized RMS displacement = 9.159054E-02
Number of optimization iterations = 260
Rotation matrix =
1.000000E+000 -5.511982E-007 6.848050E-006
5.511890E-007 1.000000E+000 1.344858E-006
-6.848051E-006 -1.344854E-006 1.000000E+000
Translation = 9.141438E-005 2.488683E-005 1.195560E-002
```

Finished writing file results\ m1.40.f.case
Finished writing file results\ m1.40.case
Increment end
CPU wall for increment 40 = 00:00:02.05, since start = 00:01:43.27
Layer end

Total number of equilibrium iterations: 79

Mesh preview volume = 4487.6249999999
Activated volume = 4487.6249999999
Activated percentage = 100.0000000000

Finished writing file \ m1.f.case
Finished writing file \ m1.case

Analysis completed

***************
16 Warnings
***************
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.
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.

![Figure 12.3: Structure type results](image)

These values correspond as follows:
-1 - Unsupported Elements
0 - Build plate
1 - Powder
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. 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.
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/((mm²).°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:

```plaintext
$ pan -b 13.final
```

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

```plaintext
$ 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:

```plaintext
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\ t13.final_171.case
```
Finished writing file results\ t13_final.171_c.case
Increment end
CPU wall for increment 171 = 00:00:00.01, since start = 00:00:14.56
Finished writing file .\ t13_final.case
Finished writing file .\ t13_final_c.case

Analysis completed

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

CPU wall  = 00:00:14.62
CPU total = 00:00:31.59

Peak RAM used for this process = 35,984 kB

END Autodesk Netfabb Local Simulation

Actual CPU times will differ. The warning is a time adjustment indicator.

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:

    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\ t13_history.171.case
    Finished writing file results\ t13_history.171_c.case
    Increment end
    CPU wall for increment 171 = 00:00:00.01, since start = 00:00:15.84
    Finished writing file .\ t13_history.case
    Finished writing file .\ t13_history_c.case

    Analysis completed

*****************************
  1 Warning
*****************************
CPU wall = 00:00:15.89
CPU total = 00:00:34.07

Peak RAM used for this process = 35,072 kB

END Autodesk Netfabb Local Simulation

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 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:
EXAMPLE 14. MULTISCALE LACK OF FUSION AND HOTSPOT PREDICTION

- 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 x 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 end of the log file will look as follows:

```
Thermal PRM input file  = 14-fine-lfus
Initial temperatures:
25.0000000000000
300.0000000000000
600.0000000000000
Lack of fusion temperatures:
1290.00000000000
1350.00000000000
Hot spots temperatures:
2200.0000000000
2600.0000000000
3000.0000000000
X size = 0.500000000000000
```
EXAMPLE 14. MULTISCALE LACK OF FUSION AND HOTSPOT PREDICTION

Y size = 0.500000000000000
Number of layers = 5

Reading input file 14-fine-lfus.in
Generating new table file: LFUS.prm

Generating threshold fraction data...

Running interlayer temperature 25.0000000000000...
Running interlayer temperature 300.0000000000000...
Running interlayer temperature 600.0000000000000...

Successfully generated prm file LFUS.prm
CPU wall = 00:11:23.85

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 /l 25 300 600 1290 1350 /o 2200 2600 3000 /d 0.5 0.5 5 > hotspot.out

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

Thermal PRM input file = 14-fine-hotspot
Initial temperatures:
25.0000000000000
300.0000000000000
600.0000000000000
Lack of fusion temperatures:
1290.0000000000000
1350.0000000000000
Hot spots temperatures:
2200.0000000000000
2600.0000000000000
3000.0000000000000
X size = 0.500000000000000
Y size = 0.500000000000000
Number of layers = 5

Reading input file 14-fine-hotspot.in
Generating new table file: hotspot.prm

Generating threshold fraction data...

Running interlayer temperature 25.0000000000000...
Running interlayer temperature 300.0000000000000...
Running interlayer temperature 600.0000000000000...

Successfully generated prm file hotspot.prm
CPU wall = 00:19:37.33
EXAMPLE 14. MULTISCALE LACK OF FUSION AND HOTSPOT PREDICTION

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:

inc = 35 time = 3352.0313 iter = 1 eps = 0.10573E+03
inc = 35 time = 3352.0313 iter = 2 eps = 0.17223E-12
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
CPU wall for increment 35 = 00:00:00.32, since start = 00:00:15.58
Layer end

Mesh preview volume = 761.06250000000
Activated volume = 761.06250000000
Activated percentage = 100.00000000000

Finished writing file .\14-lfus.case
Finished writing file .\14-lfus_c.case

Analysis completed

CPU wall for printing = 00:00:07.08
CPU wall = 00:00:15.63
CPU total = 00:00:31.77

Peak RAM used for this process = 102,392 kB

END Autodesk Netfabb Local Simulation

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:

inc = 35 time = 3880.5469 iter = 1 eps = 0.31925E+03
inc = 35 time = 3880.5469 iter = 2 eps = 0.55398E-12
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
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.
EXAMPLE 14. MULTISCALE LACK OF FUSION AND HOTSPOT PREDICTION

(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.

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.
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 over 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 & 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.

![Autogenerated mesh](image)

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$^2$)$^\circ$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$^\circ$C over a half an hour, held at 899$^\circ$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:

```bash
$ pan -b ht_bench_t
```

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

```bash
$ 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
Furnace temperature = 25.000000
  inc = 58 time = 149724.69 iter = 1 eps = 0.32805E+01
  inc = 58 time = 149724.69 iter = 2 eps = 0.12222E+00
  inc = 58 time = 149724.69 iter = 3 eps = 0.23352E-02
Finished writing file results\ ht_bench_t_58.case
Writing record: 6, time: 149724.687500000
Increment end
CPU wall for increment 58 = 00:00:00.36, since start = 00:00:23.05

Mesh preview volume = 9347.0000000000
Activated volume = 9347.0000000000
Activated percentage = 100.0000000000

Finished writing file .\ ht_bench_t.case

Analysis completed
```

**********

2 Warnings

**********
EXAMPLE 15. THERMO-MECHANICAL PROCESSING & HEAT TREATMENT MODELING

*****
1 Critical warning
*****

CPU wall for heat treatment = 00:00:02.10
CPU wall = 00:00:23.11
CPU total = 00:00:51.76

Peak RAM used for this process = 113,072 kB

END Autodesk Netfabb Local Simulation

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
Furnace temperature = 25.000000
inc = 60 time = 149724.69 iter = 1 eps = 0.20353E+07
inc = 60 time = 149724.69 iter = 2 eps = 0.40286E-08
Finished writing file results\ ht_bench_m_60_f.case
Finished writing file results\ ht_bench_m_60.case
Increment end
CPU wall for increment 60 = 00:00:00.69, since start = 00:00:48.54
CPU wall for heat treatment = 00:00:07.85

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

inc = 61 time = 199724.69 iter = 1 eps = 0.82766E+05
inc = 61 time = 199724.69 iter = 2 eps = 0.69522E-09

Optimizing rigid body motion...
Initial RMS displacement = 1.530203E-01
Optimized RMS displacement = 1.433539E-01
Number of optimization iterations = 216
Rotation matrix =
\[
\begin{bmatrix}
1.000000E+000 & 1.198705E-007 & -7.517468E-008 \\
-1.198704E-007 & 1.000000E+000 & 1.899292E-006 \\
7.517491E-008 & -1.899292E-006 & 1.000000E+000
\end{bmatrix}
\]
Translation =
\[-1.369874E-003 \quad 6.043220E-004 \quad 5.347694E-002\]

Finished writing file results\htbench_m_61_f.case
Finished writing file results\htbench_m_61.case
Increment end
CPU wall for increment 61 = 00:00:00.59, since start = 00:00:49.14

Total number of equilibrium iterations: 138
Mesh preview volume = 9347.000000000000
Activated volume = 9347.000000000000
Activated percentage = 100.000000000000

Finished writing file .\htbench_m_f.case
Finished writing file .\htbench_m.case
Analysis completed

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

**************************
1 Critical warning
**************************

CPU wall for substrate removal = 00:00:00.64
CPU wall = 00:00:49.19
CPU total = 00:02:14.62

Peak RAM used for this process = 373,368 kB

END Autodesk Netfabb Local Simulation

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

Figures 15.2 shows the temperatures during the simulated heat treatment.
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.
EXAMPLE 15. THERMO-MECHANICAL PROCESSING & HEAT TREATMENT MODELING

(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.

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.

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$^2$)$^\circ$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$^\circ$C over a half an hour, held at 899$^\circ$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$^\circ$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
Furnace temperature = 25.000000

inc =  58 time = 149724.69  iter =  1 eps =  0.32805E+01
inc =  58 time = 149724.69  iter =  2 eps =  0.12222E+00
inc =  58 time = 149724.69  iter =  3 eps =  0.23352E-02

Finished writing file results\\ ht_bench_t.out

Increment end
CPU wall for increment 58 = 00:00:00.38, since start = 00:00:23.64

Mesh preview volume = 9347.000000000000
Activated volume = 9347.000000000000
Activated percentage = 100.000000000000

Finished writing file .\\ ht_bench_t.case

Analysis completed

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

*********************************************
1 Critical warning
*********************************************

CPU wall for heat treatment = 00:00:02.00
CPU wall = 00:00:23.70
CPU total = 00:00:55.09

Peak RAM used for this process = 118,608 kB

END Autodesk Netfabb Local Simulation

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
Furnace temperature =   25.000000
  inc =  60 time =  149724.69  iter =  1 eps =  0.20353E+07
  inc =  60 time =  149724.69  iter =  2 eps =  0.39157E-08
Finished writing file results\ht_bench_m_60_f.case
Finished writing file results\ht_bench_m_60.case
Increment end
CPU wall for increment 60 = 00:00:01.39, since start = 00:01:06.94
CPU wall for heat treatment = 00:00:10.38

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

  inc =  61 time =  199724.69  iter =  1 eps =  0.82766E+05
  inc =  61 time =  199724.69  iter =  2 eps =  0.66979E-09

Optimizing rigid body motion...
Initial RMS displacement =  1.530203E-01
Optimized RMS displacement =  1.433539E-01
Number of optimization iterations =  216
Rotation matrix =
EXAMPLE 16. HEAT TREATMENT MODELING USING THE RESTART CAPABILITIES

\[
\begin{pmatrix}
1.000000E+000 & 1.198705E-007 & -7.517468E-008 \\
-1.198704E-007 & 1.000000E+000 & 1.899292E-006 \\
7.517491E-008 & -1.899292E-006 & 1.000000E+000
\end{pmatrix}
Translation = -1.369874E-003 6.043220E-004 5.347694E-002

Finished writing file results \ ht_bench_m_61_f.case
Finished writing file results \ ht_bench_m_61.case
Increment end
CPU wall for increment 61 = 00:00:00.86, since start = 00:01:07.80

-----------------------------------------------
Total number of equilibrium iterations: 138
Mesh preview volume = 9347.00000000000
Activated volume = 9347.00000000000
Activated percentage = 100.00000000000

Finished writing file . \ ht_bench_m_f.case
Finished writing file . \ ht_bench_m.case

Analysis completed

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

***************************
1 Critical warning
***************************

CPU wall for substrate removal = 00:00:01.13
CPU wall = 00:01:08.07
CPU total = 00:02:42.67

Peak RAM used for this process = 379,880 kB

END Autodesk Netfabb Local Simulation

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 =  169724.69 iter =  1 eps =  0.26334E+01
inc =  57 time =  169724.69 iter =  2 eps =  0.42881E-01
inc =  57 time =  169724.69 iter =  3 eps =  0.46977E-03

Finished writing file results'\ ht_bench_t_restart_57.case
Writing record: 5, time: 169724.687500000
Increment end

CPU wall for increment 57 = 00:00:00.39, since start = 00:00:09.45

Mesh preview volume = 9347.00000000000
Activated volume = 9347.00000000000
Activated percentage = 100.00000000000

Finished writing file \ ht_bench_t_restart.case

Analysis completed

****************************************************
1 Critical warning
****************************************************

CPU wall for heat treatment = 00:00:01.83
CPU wall = 00:00:09.50
CPU total = 00:00:11.80

Peak RAM used for this process = 111,936 kB

END Autodesk Netfabb Local Simulation

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 step # 5
Heat treatment time = 41600.000
Furnace temperature = 25.000000

inc =  59 time =  169724.69 iter =  1 eps =  0.18048E+07
inc =  59 time =  169724.69 iter =  2 eps =  0.37577E+03
inc =  59 time =  169724.69 iter =  3 eps =  0.11876E+03
inc =  59 time =  169724.69 iter =  4 eps =  0.21538E+00
inc =  59 time =  169724.69 iter =  5 eps =  0.16486E-05

Finished writing file results'\ ht_bench_m_restart_59_f.case
Finished writing file results\ht_bench_m_restart_59.case
Increment end
CPU wall for increment 59 = 00:00:01.69, since start = 00:00:18.16
CPU wall for heat treatment = 00:00:07.33

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

\begin{verbatim}
   inc =  60  time =  219724.69  iter =  1  eps =  0.37665E+05
   inc =  60  time =  219724.69  iter =  2  eps =  0.35710E-09
\end{verbatim}

Optimizing rigid body motion...
Initial RMS displacement = 1.610442E-01
Optimized RMS displacement = 1.518741E-01
Number of optimization iterations = 269

\begin{verbatim}
Rotation matrix =
  1.000000E+000  1.348451E-006  1.130048E-006
  -1.348452E-006  1.000000E+000  1.115458E-006
  -1.130047E-006 -1.115459E-006  1.000000E+000
Translation = -2.702220E-003  8.078157E-004  5.346423E-002
\end{verbatim}

Finished writing file results\ht_bench_m_restart_60_f.case
Finished writing file results\ht_bench_m_restart_60.case
Increment end
CPU wall for increment 60 = 00:00:00.55, since start = 00:00:18.72

-----------------------------------------------
Total number of equilibrium iterations: 36
-----------------------------------------------

Mesh preview volume = 9347.00000000000
Activated volume = 9347.00000000000
Activated percentage = 100.00000000000

Finished writing file .\ht_bench_m_restart_f.case
Finished writing file .\ht_bench_m_restart.case

Analysis completed

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

********************************************************************************
  2 Critical warnings
********************************************************************************

CPU wall for substrate removal = 00:00:00.61
CPU wall = 00:00:18.78
CPU total = 00:00:49.06

Peak RAM used for this process = 366,032 kB

END Autodesk Netfabb Local Simulation

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.
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 1 minute 35 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/mm² 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:
EXAMPLE 17. THERMAL MODELING USING ADVANCED CONVECTION BOUNDARY CONDITIONS

- Global Convection - *CONV = 5 W/mm² 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/mm² K. *PBCT are summed with the *CONV value, creating a total convective loss at the top of the part of 10 W/mm² 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 = 150 W/mm² 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 = 125 W/mm² 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/mm² K while the thin sections, which loose heat more rapidly, will have a heat flux of 20 W/mm² 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.
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.
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 temperatures 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 radius 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](image)

**18.2 Running Netfabb Simulation**

**18.2.1 Thermal Analysis**

To run the model, from a command line run:

```bash
$ 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 = 00:00:01.58, since start = 00:00:40.91
  inc = 34 time = 11921.886 iter = 1 eps = 0.60531E+03
  inc = 34 time = 11921.886 iter = 2 eps = 0.11601E-11
Finished writing file results \ STLH_t.34.case
Writing record: 2, time: 11921.8855986984
Increment end
CPU wall for increment 34 = 00:00:00.67, since start = 00:00:41.58
Layer end

Mesh preview volume = 17181.8507391381
Activated volume = 16969.6503833795
Activated percentage = 98.7649738146357

Finished writing file \ STLH_t.case

Analysis completed

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

*******************************************************************************
  1 Critical warning
*******************************************************************************

CPU wall for printing = 00:00:23.39
CPU wall = 00:00:41.63
CPU total = 00:01:46.81

Peak RAM used for this process = 223,880 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

inc = 38 time = 211921.89 iter = 1 eps = 0.36883E+03
inc = 38 time = 211921.89 iter = 2 eps = 0.17038E-09

Optimizing rigid body motion...
Initial RMS displacement = 1.185526E-01
Optimized RMS displacement = 8.882039E-02
Number of optimization iterations = 305
Rotation matrix =
\[
\begin{bmatrix}
1.000000E+00 & 3.209669E-005 & 2.488942E-005 \\
-3.211007E-005 & 9.999999E-001 & 5.375605E-004 \\
-2.487216E-005 & -5.375613E-004 & 9.999999E-001
\end{bmatrix}
\]
Translation = 4.584022E-003 1.309558E-002 6.933183E-002

Finished writing file results\ STLH_m.38.f.case
Finished writing file results\ STLH_m.38.case
Increment end
CPU wall for increment 38 = 00:00:02.65, since start = 00:01:32.83
Layer end

Total number of equilibrium iterations: 85

Mesh preview volume = 17181.8507391381
Activated volume = 16969.6503833795
Activated percentage = 98.7649738146357

Finished writing file .\ STLH_m.f.case
Finished writing file .\ STLH_m.case

Analysis completed

**************************
36 Warnings
**************************

**************************
3 Critical warnings
**************************

CPU wall for support removal = 00:00:02.70
CPU wall = 00:01:32.88
CPU total = 00:05:18.27
Peak RAM used for this process = 1,200,364 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 = 222,600
Number of triangles = 74,200
Finished reading Lattice.stl

Equivalencing vertices
Number of unique vertices = 35,972
Finished vertex equivalencing

Original STL volume = 149.531685569228
Seeding STL vertices with max length 8.00000000000000...
Number of seeded points = 35,972

Getting Delaunay triangulation for 35972 points...

Number of tetrahedrons = 248850
Wall time for tetrahedralization = 0.764357

Alpha radius = 10.00000000000000
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 = 93,636
Number of triangles = 31,212
Finished reading Support1_Solid.stl

Equivalencing vertices
Number of unique vertices = 14,276
Finished vertex equivalencing

Seeding STL vertices with max length 12.0000000000000...
Number of seeded points = 14,332

Getting Delaunay triangulation for 14332 points...

Number of tetrahedrons = 84954
Wall time for tetrahedralization = 0.208862

Alpha radius = 15.0000000000000
Filtering 604 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 = 17,937
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
Seeding STL vertices with max length 4.000000000000000...
Number of seeded points = 5120

Getting Delaunay triangulation for 5120 points...
Number of tetrahedrons = 28056
Wall time for tetrahedralization = 0.0647566

Alpha radius = 5.000000000000000
Filtering 451 tetrahedrons...
Number of hull triangles = 2964
Finished writing binary STL file Support2_0Thickness_Loose_concavity.stl
Homogenized STL volume = 924.134388621193
Volume fraction = 0.1100964

Homogenizing STL 5...
Reading Support3_0Thickness_Fine.stl
Reading in native format...
Binary STL file
Bounding box:
1.731400E+01 <= x <= 3.065600E+01
-7.656000E+00 <= y <= 5.682000E+00
0.000000E+00 <= z <= 1.937400E+01

Number of vertices = 73,137
Number of triangles = 24,379
Finished reading Support3_0Thickness_Fine.stl

Equivalencing vertices
Number of unique vertices = 19,241
Finished vertex equivalencing

Calculating surface normals
Original STL surface area = 1113.08966895905
Original STL volume = 166.963450343858
Seeding STL vertices with max length 4.00000000000000...
Number of seeded points = 19,409

Getting Delaunay triangulation for 19409 points...

Number of tetrahedrons = 114150
Wall time for tetrahedralization = 0.312326

Alpha radius = 5.00000000000000
Filtering 2006 tetrahedrons...
Number of hull triangles = 7888
Finished writing binary STL file Support3_0Thickness_Fine_concavity.stl
Homogenized STL volume = 933.467339324020
Volume fraction = 0.1788637

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 - 0.178

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 0.178. The Solid and Fine supports exhibit an equivalent trend and value of distortion, while the Loose lattice type support shows roughly 25% more distortion as the other supports.

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. This shows that these disparate support types all exhibit similar levels of failure.
Figure 18.3: STL homogenization example displacement results
Figure 18.4: STL homogenization example support failure results
Example 19

Custom Buildplate Geometry in Part Scale Powder Bed 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.

19.1 Problem Description

An Inconel® 625 test geometry is constructed on top of an Inconel 625 cylindrical build plate, using generic laser powder bed fusion processing parameters. Both the part and build plate geometries are imported in the analysis through STL files and both are automatically meshed within Netfabb Simulation. The build plate STL is assigned using the *STLM card by setting the configuration id=2. The PRM number is not used so 1 is used as a dummy value. The Material is the same as the PRM file so the Material ID=2. No homogenization is used for the build plate so the Volume Fraction is set to 1. The *STLM card settings here then are:

*STLM
2, 1, 1, 1.0

Constant build plate heating at 120 °C., enabled by the *PBLR card, is used to mitigate distortion. The mesh and support type is shown in Figures 19.1.

As in previous examples, first a time incremental thermal analysis is performed to ascertain temperature history of the part through the manufacturing process and post-processing steps. A subsequent time incremental mechanical analysis is then completed to determine mechanical response.
19.2 Running Netfabb Simulation

19.2.1 Thermal Analysis

To run the model, from a command line run:

```
$ pan -b CanonOnCylinder1_thermal
```

The analysis progress is written on file `CanonOnCylinder1_thermal.out`. To check progress run:

```
$ tail CanonOnCylinder1_thermal.out
```

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

```
inc = 44 time = 14849.417 iter = 1 eps = 0.38052E+00
inc = 44 time = 14849.417 iter = 2 eps = 0.20398E-12
Finished writing file results\ CanonOnCylinder1_thermal.case
Finished writing file results\ CanonOnCylinder1_thermal_c.case
Writing record: 2, time: 14849.4166881167
Increment end
CPU wall for increment 44 = 00:00:00.94, since start = 00:01:04.85
Layer end

Mesh preview volume = 26925.5993719995
```
Activated volume = 26925.5993719995
Activated percentage = 100.000000000000

Finished writing file .\ CanonOnCylinder1_thermal.case
Finished writing file .\ CanonOnCylinder1_thermal_c.case

Analysis completed

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

CPU wall for printing = 00:00:48.67
CPU wall = 00:01:04.90
CPU total = 00:03:20.08

Peak RAM used for this process = 401,868 kB

END Autodesk Netfabb Local Simulation

Actual CPU times will differ.

19.2.2 Quasi-Static Mechanical Analysis

Run the mechanical analysis from the command line:

    $ pan -b CanonOnCylinder1_mechanical

    After the analysis completes, the last few lines of the output file CanonOnCylinder1_mech.out should be similar to the following:

----------------------------------
Substrate removal time increment
----------------------------------
    inc = 46 time = 114849.42 iter = 1 eps = 0.54555E+04
    inc = 46 time = 114849.42 iter = 2 eps = 0.20850E-08

Optimizing rigid body motion...
Initial RMS displacement = 2.573829E-01
Optimized RMS displacement = 1.310762E-01
Number of optimization iterations = 305
Rotation matrix =
    9.999858E-01  -2.274892E-04  5.315012E-03
    2.012156E-04  9.999878E-01  4.943296E-03
    -5.316072E-03  -4.942157E-03  9.999737E-01
Translation = -6.290836E-02 -4.968993E-02 -3.091511E-01

Finished writing file results\ CanonOnCylinder1_mechanical_f.case
Finished writing file results\CanonOnCylinder1_mechanical.case
Increment end
CPU wall for increment 46 = 00:00:02.87, since start = 00:02:13.07
Layer end

Total number of equilibrium iterations: 91

Mesh preview volume = 26925.5993719995
Activated volume = 26925.5993719995
Activated percentage = 100.000000000000

Signal tag 604A
*** CRITICAL WARNING: 1
Recoater Interference Detected at 1 layer group. Minimum clearance of 48.0257034301701 at height 20.4000000000000 mm.

Finished writing file .\CanonOnCylinder1_mechanical_f.case
Finished writing file .\CanonOnCylinder1_mechanical.case

Analysis completed

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

******************************
1 Critical warning
******************************

CPU wall for substrate removal = 00:00:02.93
CPU wall = 00:02:13.12
CPU total = 00:06:41.67

Peak RAM used for this process = 1,186,964 kB

END Autodesk Netfabb Local Simulation

Actual CPU times will differ.

19.3 Results

Figures 19.2 shows the computed final distortion from the mechanical analysis after part construction and cool down, and after part is removed from the buildplate.
EXAMPLE 19. CUSTOM BUILDPLATE GEOMETRY IN PART SCALE POWDER BED MODELING

Figure 19.2: Distortion results [mm]

(a) After part construction and cool down

(b) After the component is released from the buildplate
Example 20

6 Axis 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.

20.1 Problem Description

This example simulates the 6 Axis Directed Energy Deposition (DED) construction of a two bead wide, 3 layer high Ti-6Al-4V component on a radial Ti-6Al-4V component. The radial component is shown in Figure 20.1 while the laser path is shown in Figure ?? upon a generic substrate. Note that 6 axis DED laser path (.lsr) files can be imported and viewed in the Simulation for Netfabb Software, however the simulations must still be performed at the command line. The radius of the melt pool is 2 mm, its power is 750 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 resulting mesh is shown in Figure ??.

20.2 Running Netfabb Simulation

20.2.1 Thermal Analysis

Run the analysis from the command line:

    $ pan -b 6axis_thermal

After the analysis completes, the last few lines of the output file 6Axis_thermal.out should be similar to the following:

Increment end
CPU wall for increment 173 = 00:00:00.45, since start = 00:02:51.11
   inc = 174 time =  1000.0000 iter = 1 eps = 0.75919E-02
   inc = 174 time =  1000.0000 iter = 2 eps = 0.45554E-02
   inc = 174 time =  1000.0000 iter = 3 eps = 0.65062E-06
Finished writing file results\ 6Axis_thermal.case
Writing record: 91, time: 1000.000000000000
Increment end
EXAMPLE 20. 6 AXIS DIRECTED ENERGY DEPOSITION

CPU wall for increment 174 = 00:00:00.49, since start = 00:02:51.61
Finished writing file .\6Axis_thermal.case

Analysis completed

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

CPU wall    = 00:02:51.66
CPU total   = 00:11:12.98

Peak RAM used for this process = 117,232 kB

END Autodesk Netfabb Local Simulation

20.2.2 Mechanical Analysis

Run the analysis from the command line:

$ pan -b 6axis_mechanical

After the analysis completes, the last few lines of the output file 6axis_mechanical.out should be similar to the following:

Increment end
CPU wall for increment 169 = 00:00:01.34, since start = 00:04:09.18
   inc =  170 time =  1000.0000  iter =  1 eps =  0.24387E+03
   inc =  170 time =  1000.0000  iter =  2 eps =  0.19510E+03
   inc =  170 time =  1000.0000  iter =  3 eps =  0.64093E-09
Finished writing file results\ 6Axis_mechanical.case
Increment end
CPU wall for increment 170 = 00:00:01.25, since start = 00:04:10.43

----------------------
*COOL time increment
----------------------
HTOR is being set to zero***
   inc =  171 time =   1100.0000  iter =  1 eps =  0.64174E+03
   inc =  171 time =   1100.0000  iter =  2 eps =  0.51340E+03
   inc =  171 time =   1100.0000  iter =  3 eps =  0.64679E-09
Finished writing file results\ 6Axis_mechanical.case
Increment end
CPU wall for increment 171 = 00:00:01.35, since start = 00:04:11.79

------------------------------------------------------
Total number of equilibrium iterations: 548
Finished writing file \ 6Axis_mechanical.case

Analysis completed

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

CPU wall for cooldown = 00:00:01.48
CPU wall     = 00:04:11.92
CPU total     = 00:16:27.36

Peak RAM used for this process = 495,720 kB

END Autodesk Netfabb Local Simulation
EXAMPLE 20. 6 AXIS DIRECTED ENERGY DEPOSITION

Figure 20.1: 6 Axis DED example part and path.

(a) 6 Axis DED example geometry

(b) 6 Axis DED example path
Figure 20.2: 6 axis DED mesh
20.3 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 20.3. Post process distortion and a sample stress result is shown in Figure 20.4.
Figure 20.3: Temperature results (°C) at two sample increments.
EXAMPLE 20. 6 AXIS DIRECTED ENERGY DEPOSITION

Figure 20.4: Sample post process mechanical results

(a) Post Process distortion results, warped 1X

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