Netfabb Local Simulation

Keyword User Manual

Version 2021.1

©September 4, 2020

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 Overview of the Manual and Netfabb Simulation **7**

1.1 Outline 7

1.2 Getting Started 7

1.3 Recommended minimum system requirements 7

1.3.1 Setting up the Netfabb Simulation environmental variables 8

1.4 Linux systems Shell and Intel® MKL Environment variables 8

1.5 Netfabb Simulation Environment variables 8

1.6 Netfabb Simulation Capabilities 9

1.6.1 Powder-Bed Part-Level 9

1.6.2 Powder-Bed Fine-Scale 12

1.6.3 Directed Energy Deposition 13

1.7 Pre-processing 14

1.8 Post-processing 15

2 Execution and File Notation **17**

2.1 Execution 17

2.1.1 From the file browser (Windows only) 17

2.1.2 From the Windows Command Line or Linux terminal 17

2.1.3 PRM file generation execution . . . 19
2.1.4 Interrogating prm files to determine prm type . . . 21
2.1.5 Automatic Calibration of prm files . . . 21
2.2 File extensions . . . 22
2.2.1 Input Files . . . 22
2.2.2 Output Files . . . 22

3 Keyword Listing **25**

3.1 Variable Type Notation . . . 25
3.2 System checks . . . 25
3.3 Input/Output . . . 25
3.3.1 Part Level Stress Output . . . 31
3.4 Analysis control . . . 33
3.5 Element Activation Strategies . . . 39
3.5.1 6 Axis DED simulations . . . 41
3.6 Part-Level Powder-Bed Modeling . . . 47
3.7 Material Property Definition . . . 65
3.8 Boundary Conditions . . . 72
3.8.1 Temperature Boundary Conditions . . . 73
3.8.2 Convection Boundary Conditions . . . 74
3.8.3 Heat Source Boundary Conditions . . . 78
3.8.4 Mechanical Constraints . . . 83
3.9 Additional Options . . . 87
3.10 Support Structure Modeling . . . 92

3.11 Heat treatment modeling 96

4 Post processing tools **99**

5 Units **105**

6 Common errors and warnings **107**

6.1 Errors 107

6.1.1 Autodesk Netfabb Local Simulation LT Errors 107

6.1.2 Runtime errors 107

6.1.3 Licensing errors 108

6.1.4 Netfabb Simulation LT errors 108

6.1.5 Basic input file errors 109

6.1.6 Thermal-mechanical input file mismatch errors 111

6.1.7 PRM generation errors 111

6.1.8 Part scale geometry errors 112

6.1.9 Element Activation Errors 114

6.1.10 Substrate specification errors 114

6.1.11 Meshing errors 115

6.1.12 Simulation settings errors 115

6.1.13 Material property errors 116

6.1.14 Boundary condition errors 117

6.1.15 Output errors 118

6.1.16 Convergence errors 118

6.1.17 Patran mesh errors 119

6.1.18 Laser file errors . . . 119
6.1.19 Heat treatment errors . . . 120
6.1.20 End of Simulation Errors . . . 121
6.2 Critical Warnings . . . 121
6.2.1 System Critical Warnings . . . 121
6.2.2 PRM File Critical Warnings . . . 122
6.2.3 STL file and Meshing Critical Warnings . . . 122
6.2.4 Convergence Critical Warnings . . . 123
6.2.5 Simulation settings Critical Warnings . . . 124
6.2.6 Heat source Critical Warnings . . . 124
6.3 Warnings . . . 125
6.3.1 Run time warnings . . . 125
6.3.2 Material property warnings . . . 125
6.3.3 Boundary condition warnings . . . 126
6.3.4 Mesh warnings . . . 126
6.3.5 Build failure warnings . . . 127
6.3.6 Heat treatment model warnings . . . 127
6.3.7 Output warnings . . . 128

Chapter 1

Overview of the Manual and Netfabb Simulation

1.1 Outline

This manual is a guide to using Netfabb Simulation by Autodesk Inc. Users may also find it helpful to read through the accompanying examples manual. Users that prefer to use the Netfabb Simulation GUI, Autodesk Simulation Utility for Netfabb, rather than directly using the input files should consult the online help repository. The present manual is divided into the following chapters:

- Getting Started: This chapter lists the recommended minimum system requirements, how to set up the environmental variables, and describes the workflow of the various Netfabb Simulation simulation types.
- Execution and File Notation: This chapter describes the process of executing an analysis. A description is given of the various output files that are generated.
- Keyword Listing: This chapter describes the various analysis cards that can be activated in the Netfabb Simulation input file. Only the first four characters of the cards are read, e.g. the input '*TITLE' is read as '*TITL'.

1.2 Getting Started

This section will briefly inform the new user how to set up the system on a new machine and outline the purpose, use, and workflow of the simulation types supported by Netfabb Simulation .

1.3 Recommended minimum system requirements

To run Netfabb Simulation simulations it is recommended that the workstation have at least:

- 14 Processing cores
- 64 GB of RAM

1.3.1 Setting up the Netfabb Simulation environmental variables

By default, Netfabb Simulation will use the maximum number of available cores. If hyper-threading is not turned off, it will use twice the number of physical cores which results in slow performance. It is best to set the maximum number of threads to the physical cores. To ensure that Netfabb Simulation properly utilizes system resources, the following steps should be followed after installation.

1. Navigate to the Netfabb Simulation environmental variable file which will be in `C:\Program Files \Autodesk \Netfabb Local Simulation 20XX \env \pan.env`
2. Right click on the pan.env file, Choose Open With - and select your preferred text editor.
3. Set `OMP_NUM_THREADS:` to the number of **Physical processing cores** of your system. If your system uses hyperthreading do not use the number of apparent cores as this will slow down performance.
4. Set `KMP_STACKSIZE:2G`
5. Save and close the file

Users may also set the `OMP_NUM_THREADS` and `KMP_STACKSIZE` variables as user or environmental variables through Windows environmental variable settings. This is required for simulations using the `pan2` command directly instead of the `pan` command, as `pan2` does not reference the pan.env file.

1.4 Linux systems Shell and Intel® MKL Environment variables

To control the number of threads use the OMP_NUM_THREADS environment variable. For example, for 16 threads type in the bash shell:

```
export OMP_NUM_THREADS=16
```

By default, Intel64 sets the limit of the number of bytes to allocate for each OpenMP* thread to 4m (megabytes). For very large systems (more than 400,000 degrees of freedom) this may cause a segmentation fault. To change this, unlimit the KMP_STACKSIZE environment variable. For example for 1Gb type in bash shell:

```
export KMP_STACKSIZE=1g
ulimit -s unlimited
```

It is recommended to edit these environment variables definitions in the /.bashrc (or /.cshrc) file.

1.5 Netfabb Simulation Environment variables

Additional environment variables for Netfabb Simulation can be set using a `pan.env` file. The variables need only to be defined if it is desired to override the default definitions. Netfabb Simulation and the auxiliary programs look in the following directories for the `pan.env` file:

```
.pan_env
~/pan_env
$panDir/pan_env
/opt/pan/pan_env
```

If no `pan.env` file is found, the default values are used. For convenience during the Netfabb Simulation installation the contents of the following file are copied to the `$panDir/pan_env/pan.env` file:

```
#
# This file contains definitions of environment variables for Pan and
# the auxiliary programs such as plotp3
#
KMP_STACKSIZE:500000000
##Uncomment and edit the following line to override the default number of threads.
#OMP_NUM_THREADS:4
```

For Windows installations, unless OMP_NUM_THREADS has been manually set either as a Windows environment variable or via the pan.env, at run time the solver determines the number of physical cores of the system and uses that value for the number of threads during simulation.

1.6 Netfabb Simulation Capabilities

The Netfabb Simulation solver performs thermo-mechanical analyses for all common AM processes. The user passes the relevant process parameters into the solver by creating an input file. The input file also allows for various features of the solver to be controlled by using the available analysis cards described in Chapter 3 "Keywords" of this manual. The output of the thermal analysis is temperature history and the output of the mechanical analysis is deformation, stress, and strain. There are several types of analyses that can be run in Netfabb Simulation . They are as follows:

1.6.1 Powder-Bed Part-Level

Netfabb Simulation Powder-Bed Part-Level analysis utilizes a 2 part multi-scale modeling approach. A detailed Process Parameter model is linked to the Part-Level model through a Process Parameter (.prm) file. Figure 1.1 illustrates the workflow. The 2 stages of the analysis are described below.

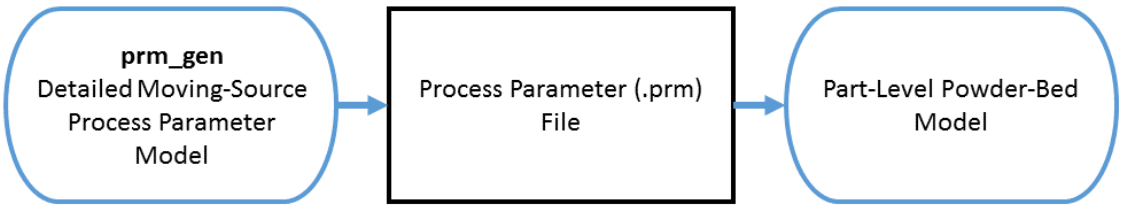


Figure 1.1: Netfabb Simulation Process Parameter (.prm) file generation.

Detailed Process Parameter Model

For each new set of process parameters, the user must generate a process parameter (.prm) file. The .prm file is an input into the Part-Level analysis that accounts for changes in the deposition settings. This type of analysis needs only to be run once for each set of parameters. Once the .prm

file is generated for a set of parameters it can repeatedly be used as an input into the Part-Level analysis. Figure 1.2 illustrates the workflow.
From version 2.91 onward, there are 3 kinds of prm files that can be created: mechanical analysis only, thermal analysis only, or both mechanical and thermal analysis. Mechanical only prm files are used to predict the part level distortion, residual stress, and stress relief heat treatment. Thermal analysis prm files are used to predict lack of fusion or hotspots during laser powder bed fusion processes. Users can also opt to include both analysis types in a single prm generation simulation. These prm generation processes require the additional inputs of Lack of Fusion temperatures, Hotspot temperatures, and interlayer temperatures.
To obtain accurate PRM files use prm_gen to run produce the .prm file for each set of process parameters and material properties

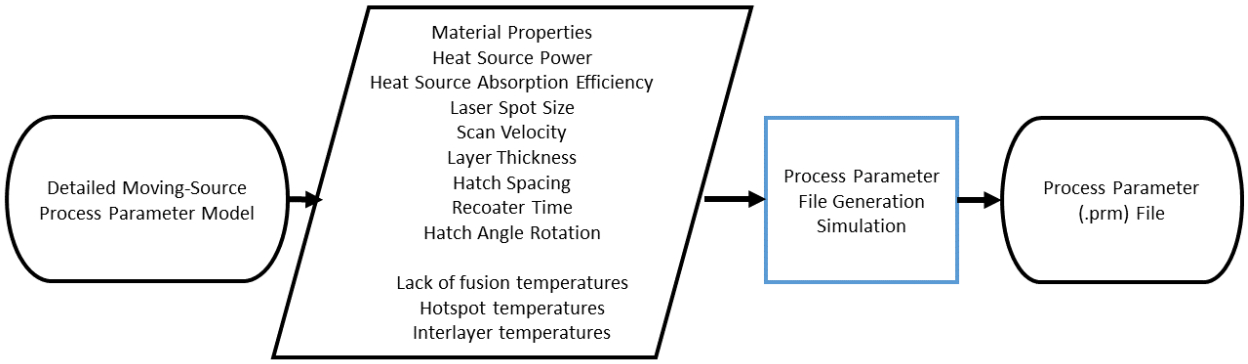


Figure 1.2: Netfabb Simulation Detailed Process Parameter Model.

Mechanical Only Powder-Bed Part-Level Model

Description: The mechanical Powder-Bed Part-Level capability of Netfabb Simulation allows for users to simulate the deposition of a component of any size or geometric complexity.
Output: The output of the mechanical analysis is interlayer temperatures, displacement, and stress. Post-process distortion and stress can be viewed both before and after support structure and removal from the build plate. Netfabb Simulation can also predict support structure failure and part interference with the recoater blade due to excessive vertical distortion.
Common Uses: The results can be used to guide decisions about part orientation, support structure placement and design, distortion based geometry compensation, and process planning.
Workflow: The approach begins by generating a Process Parameter file (known as a PRM file) based on know system Parameters. The Process Parameter files for several materials using generic process parameters settings are included with the Netfabb Simulation license. Users should generate their own PRM files for the processing conditions and materials used in their powder bed systems. The Process Parameter file is input into the Netfabb Simulation thermal solver along with information about the geometry being built. The user can choose to simulate the entire build plate, or a simplified analysis of just an individual part. The output thermal history is then input

in the Netfabb Simulation mechanical solver. Figure 1.3 illustrates the workflow for this analysis type.

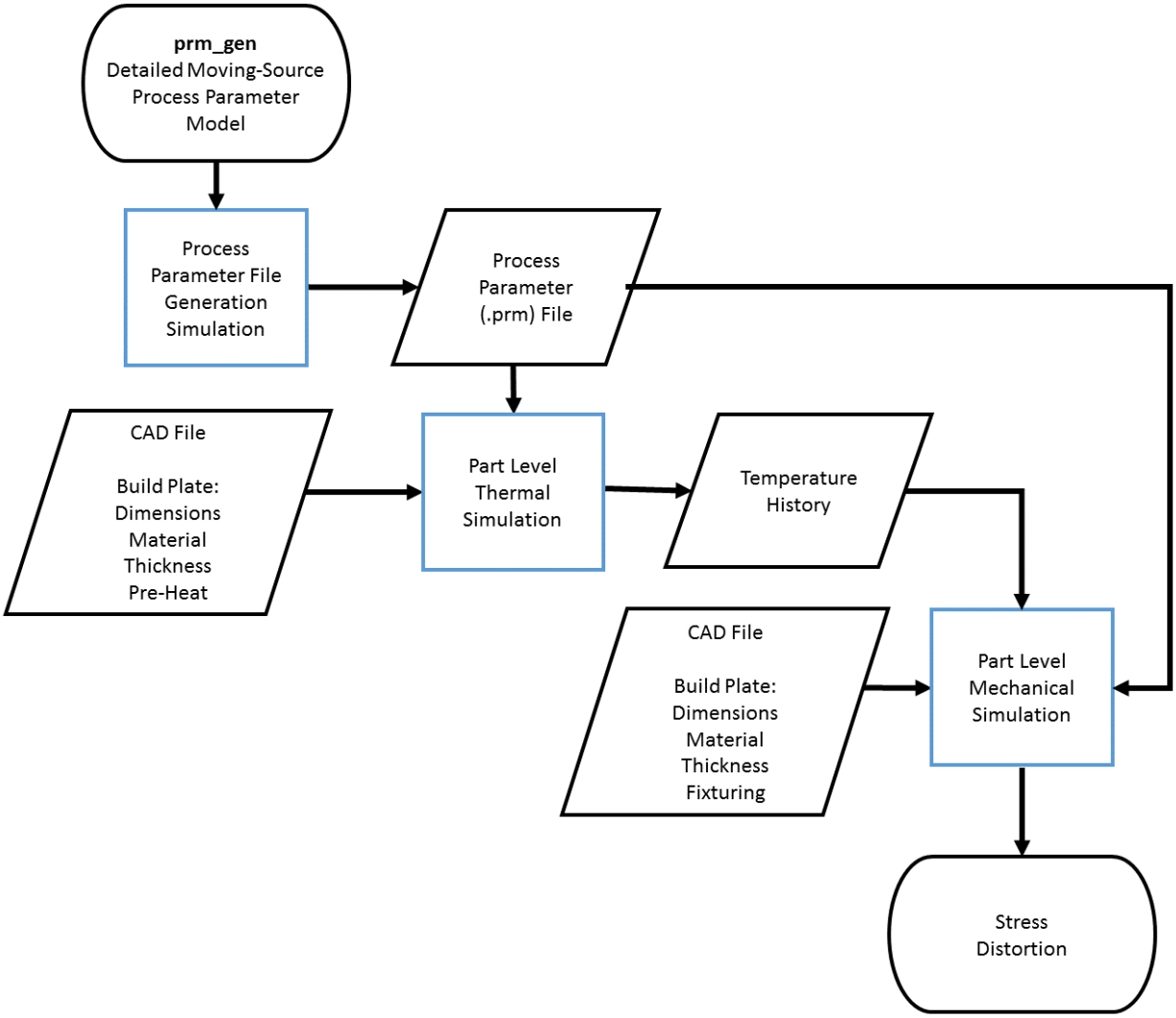


Figure 1.3: Netfabb Simulation Powder-Bed Part-Level Model.

Thermal Only Powder-Bed Part-Level Model

Description: The thermal Powder-Bed Part-Level capability of Netfabb Simulation allows for users to simulate the deposition of a component of any size or geometric complexity.
Output: The output of the thermal analysis is peak interlayer temperatures, regions exhibiting lack of fusion or hot spots.
Common Uses: The results can be used investigate the potential quality of parts produced with a certain set of processing parameters using a particular material. This can prevent costly experimental iterations before printing with a new set of processing conditions.

Workflow: First a thermal PRM file must be created. Generic lack of fusion and hotspot PRM files are included with the solver, but users are advised to produce files based upon the processing parameters they use for their LPBF systems. Then a part scale thermal analysis is performed using the PRM file and the geometry or geometries of interest. The results show where regions where lack of fusion or hotspots may be an issue using the source processing parameters. The thermal only part scale analysis is shown in Figure 1.4.

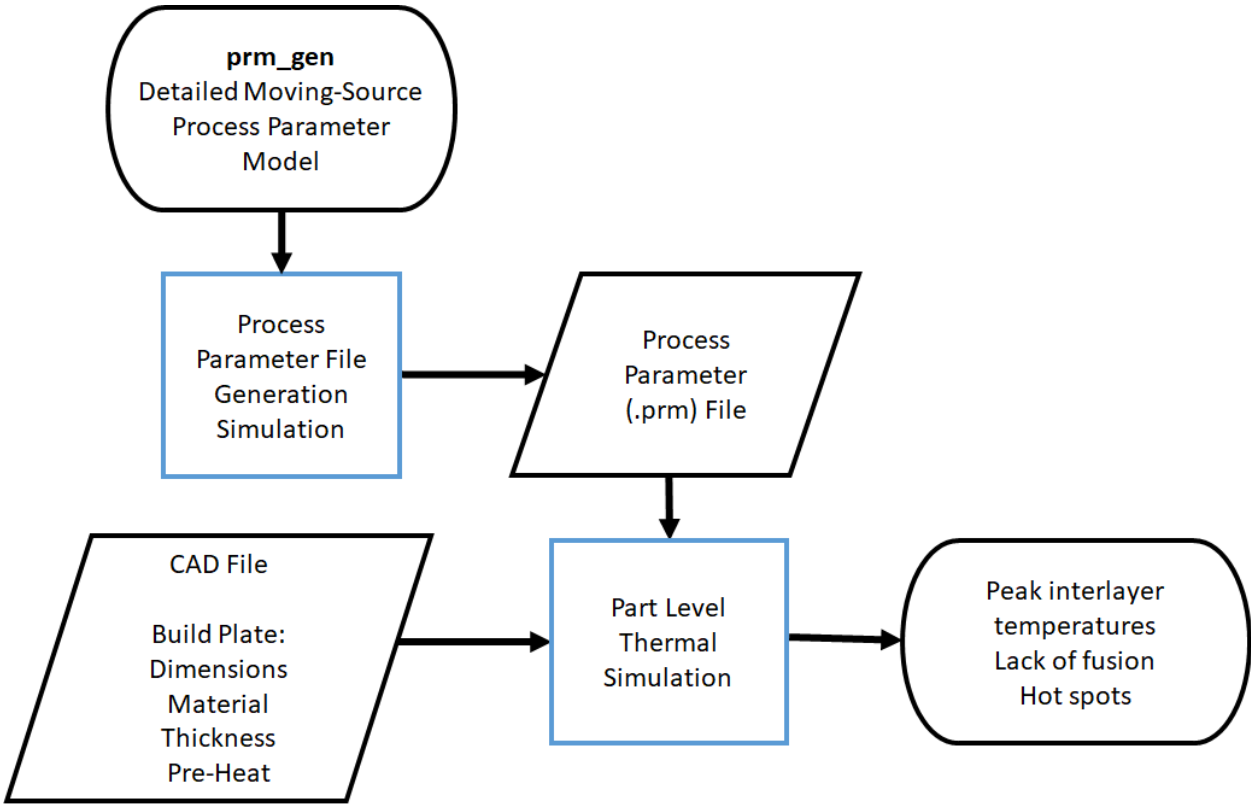


Figure 1.4: Netfabb Simulation Powder-Bed Part-Level Model.

Thermal and Mechanical Powder-Bed Part-Level Model

Users can produce PRM files with both thermal and mechanical information to drive part scale simulations. The process is identical to the sections above, where first a prm generation process is completed, then that prm is used with selected geometry or geometries. The results are predictions of lack of fusion, hotspots, distortion, and stress.

1.6.2 Powder-Bed Fine-Scale

Description: The Powder-Bed Fine-Scale capability of Netfabb Simulation allows users to simulate depositions using a full moving-source analysis.
Output: The output of the analysis is temperature, distortion, stress, and strain.
Common Uses: The results can be used to guide microstructure prediction, lack of fusion prediction, and process parameter selection.

Workflow: The approach begins by submitting known process parameters into the Netfabb Simulation thermal solver. The output thermal history is then input into the Netfabb Simulation mechanical solver. Figure 1.5 illustrates the workflow.

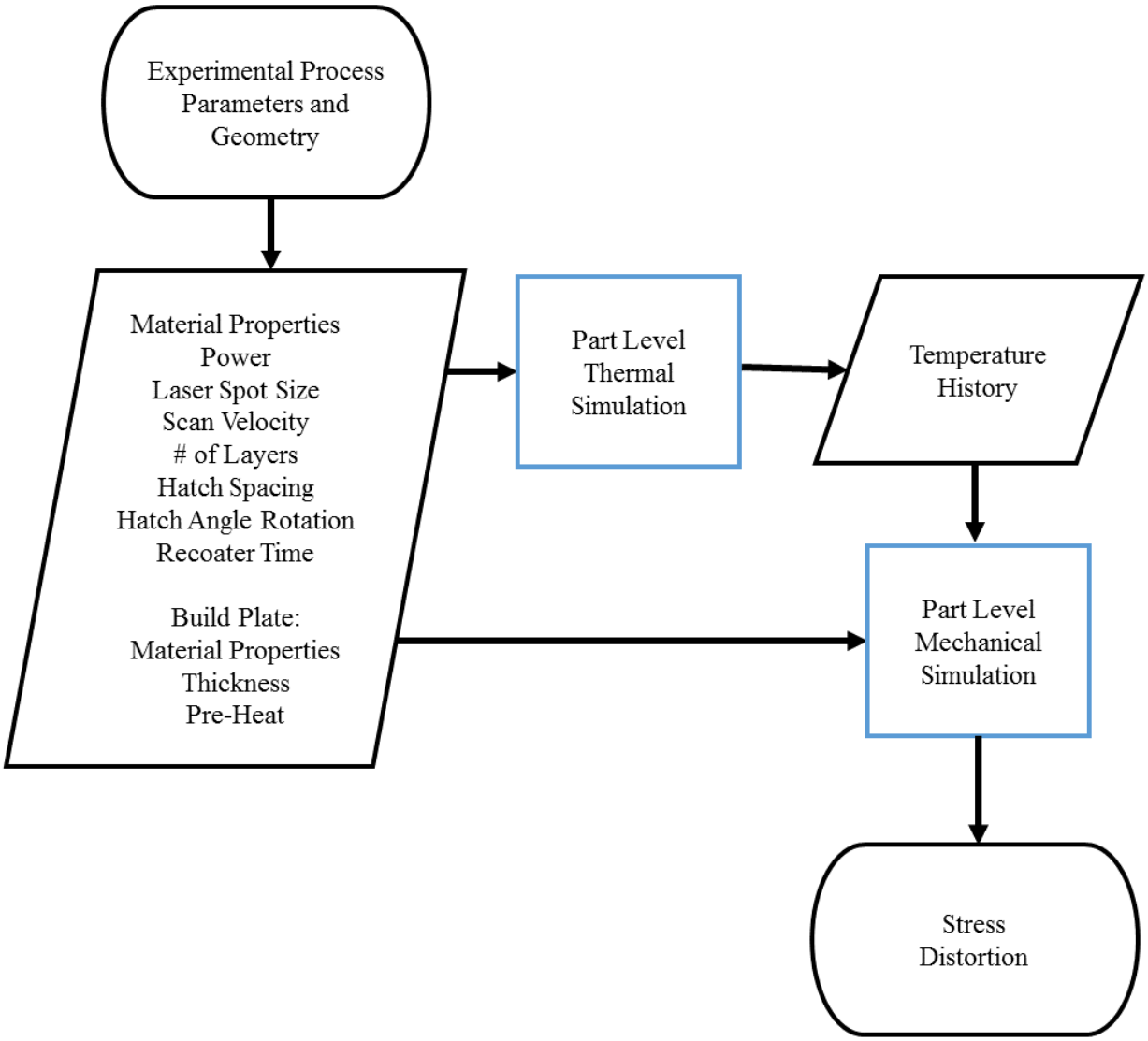


Figure 1.5: Netfabb Simulation Powder-Bed Fine-Scale Model.

1.6.3 Directed Energy Deposition

Description: The Directed Energy Deposition capability of Netfabb Simulation allows users to simulate full builds for both Powder-Fed and Wire-Fed AM processes.
Output: The output of the analysis is temperature, distortion, stress, and strain.
Common Uses: The results can be used to guide process parameter selection and path planning.
Workflow: The approach begins by submitting known process parameters into the Netfabb

Simulation thermal solver. The output thermal history is then input into the Netfabb Simulation mechanical solver. Figure 1.6 illustrates the workflow.

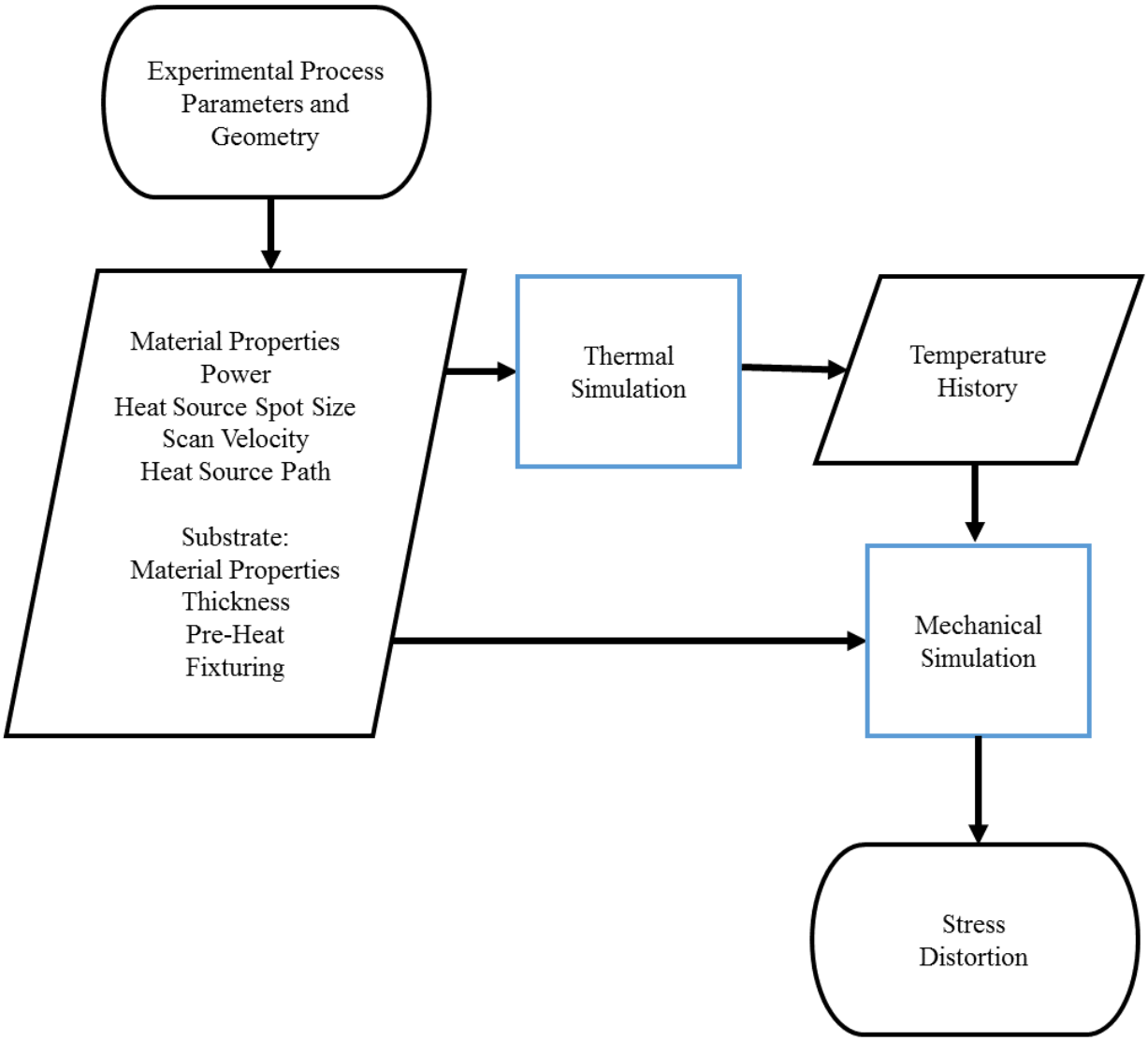


Figure 1.6: Netfabb Simulation DED Model.

1.7 Pre-processing

Pre-process mesh generation can be performed in two ways:

1. Automatically generate the mesh directly from a .stl file or heat source path using the Netfabb Simulation automatic mesh generation feature
2. Manually generate the mesh in Patran (software by MSC)

1.8 Post-processing

Supported post-processing software for the Netfabb Simulation solver:

1. Autodesk Simulation Utility for Netfabb
2. Paraview (free open-source software available at: http://www.paraview.org)
3. Patran (software by MSC)

Chapter 2

Execution and File Notation

2.1 Execution

2.1.1 From the file browser (Windows only)

During installation the *.in file type should be automatically associated with Netfabb Simulation . This allows execution by double clicking any *.in file in the file browser.
If for some reason this fails or if the end user changes the filetype association, execution of Netfabb Simulation can be performed by:
Right click the *.in file. Choose "open with" pan.exe for the file (<fname>.in) to run the analysis. Refer to Getting Started for the path to the pan.exe file and instructions on how to associate filetypes in Windows.
To edit input files:
Right click the *.in file. Choose "open with" (your choice of text editor) to edit the analysis control file (<fname>.in).
To see the execution options, in the command line type pan- h:

2.1.2 From the Windows Command Line or Linux terminal

```
$ pan -h

Wrapper for Netfabb Local Simulation execution

 Autodesk, Inc. - www.pancomputing.com

 version 2


 Synopsis
 pan [OPTIONS] [filename]

 filename is either the input file name (w/o .in) or a batch file
 (w/o .que) containing a list of input file names for batch execution.

 OPTIONS:
```

```
-b    background mode. In background mode screen
      output is redirected to fileneme.out.

-c    cost mode.
      This option generates a mesh preview, computes
      the number of layers, and terminates execution.

-f    fast cost mode.
      This option estimates the memory required for a mesh
      preview.

-e exec   define executable  different than default.
      exec is either the full path of the executable or a
      link to it. This option allows using custom (user)
      compiled versions of Netfabb Local Simulation

-h      help

-l    use .log instead of .out for output files

-m    mesh preview mode.
      This option generates a mesh preview, computes
      the number of layers, and prompts to continue execution.

-p number   set number of threads

-q    queue (batch) mode using list in filename.que

-s    silent mode (no questions if previous files exist)

-t xx   copy template mode.
    xx is the examples template number
    This option copies an examples template to the current
    directory
    If no argument xx is provided, then a description of
    all examples will be displayed

--examples    display examples manual

--user    display user manual

Examples:

Interactive mode:

  pan
```

```
  this mode will run the default version and will
             ask for an input file name

  pan filename
  this mode will run the default version and will
             use filename.in as input file

  pan filename -p 6
  this mode will run the default version and will
             use filename.in as input file, using 6 threads.

  pan --user
  display user manual

Background mode:

  pan -b filename
  this mode will run version 261 in background
             using filename.in as input file

Batch mode:

  pan -q filename
  this mode will run version 261 in background sequentially
             for all input file names listed in file filename.que

Custom mode:

  pan -e /path/mypan2 -b filename
  this mode will run custom executable /path/mypan2
  in background using filename.in as input file

Template mode:

  pan -t 01
  this mode will copy examples 01 template to current
  directory"
```

Please, note that the -v option is only available for Linux version only.

2.1.3 PRM file generation execution

PRM generation is completed using the command `prm_gen`.

```
prm_gen [/t thermal.in] [/m mechanical.in] [/i tmp1 tmp2 ...] [/l tlthresh1 tlthresh2 ...]
```

```
[/o tothresh1 tothresh2 ...] [/d xsize ysize nlayers] [/g]
```

`/t` specifies the thermal input file.
`/m` specifies the mechanical input file.
`/i` lists the interlayer temperatures to generate tabular data for. More interlayer temperatures can create more accurate predictions of lack of fusion or hot spot temperatures, especially for materials with sever non-linearities in specific heat and thermal conductivity, however additional temperatures will significantly increase run time.
`/l` lists the threshold temperatures which not reaching may result in lack of fusion.
`/o` lists the threshold temperatures which exceeding may result in hotspots.
`/d` lists the x dimension, y dimension, and number of layers for the lack of fusion and hotspot prediction prm generation process.
`/g` is used to query an already generated prm file to learn if the file can be used for mechanical simulations only, thermal simulations only, or both. `/c` engages the prm auto-calibration process.

Standard thermo-mechanical PRM files:

For standard thermo-mechanical PRM files the basic usage is:
`prm_gen thermal.in mechanical.in`
To direct the output to a logfile:
`prm_gen thermal.in mechanical.in > prm.out`
The resulting prm file may be used for part scale predictions of powder bed fusion stress and distortion.

Thermal only PRM files:

For a PRM file used only for investigating thermal behavior including peak post deposition temperatures, lack of fusion and hotspots:

```
prm_gen [/t thermal.in] [/i tmp1 tmp2 ...] [/l tlthresh1 tlthresh2 ...]

[/o tothresh1 tothresh2 ...] [/d xsize ysize nlayers]
```

Where `tmp1` etc. are interlayer temperatures, `tlthresh1` etc. are lack of fusion temperatures, `tothresh1` etc. are overheating temperatures, `xsize` and `ysize` determine the small-scale x and y bounds, and `nlayers` controls the number of layers to simulate.
This will produce a prm file that can be used for part scale predictions of lack of fusion or hotspot behavior in powder bed fusion processes.
Thermal only PRM file requirements and defaults:
For thermal only PRM generation at least one interlayer temperature and at least one lack of fusion or overheating temperature is required. By default thermal only PRM generation defaults to `xsize = 1 mm`, `ysize = 1 mm`, and `nlayers = 5`. You need only invoke these options to override the default values.
Thermal only PRM generation example:
For Inconel 625:

```
prm_gen /t inc625_thermal.in /i 25 100 300 600 /l 1290 1350 /o 2500 3000

/d 0.5 0.5 5 > inc625_thermal_prm_gen.out
```

This will create look up table values for interlayer temperatures at 25, 100, 300, and 600°C, check for volume fractions that do no reach 1290 or 1350°C (the solidus and liquidus temperatures), and check for volume fractions that exceed 2500 and 3000°C. The resulting PRM file can only be used for thermal part scale analyses.

Full thermal-mechanical PRM generation example:

```
prm_gen /t inc625_thermal.in /m inc625_mechanical.in /i 25 100 300 600 /l 1290 1350

/o 2500 3000 /d 0.5 0.5 5 > inc625_prm_gen.out
```

This will produce a PRM file than can be used for both predictions of mechanical behavior, like part scale stress and distortion, and thermal behavior, like lack of fusion and hotspots.

2.1.4 Interrogating prm files to determine prm type

If there is uncertainty whether a prm file can be used for thermal only, mechanical only, or both types of part scale simulations, users may query the prm file using the `/g` option. This option has the syntax:

```
prm_gen /g example.prm
```

2.1.5 Automatic Calibration of prm files

Users now have the option to perform a experimentally based automatic calibration of PRM files. This is useful to tailor prm files to adjust for a consistent over or under prediction, as is done using the manual prm scaling option. This tool can also be used to rapidly produce a prm file for a material not yet supported which is functionally similar to an existing validated material in your prm library.

To proceed with the calibration, first build the compliant flexible cylinder available to download here using the material and processing conditions it is desired to calibrate.

Measure the peak deformation at the center of the part. This can easily be done with calipers, a micrometer, or through scanning process. The measurement should be done with the part still on the build plate and without any heat treatment processing.

To run the calibration process the following files are required:

- Base prm file. This should be for the same alloy or the most similar alloy available. Using too dissimilar material properties will lead to a poorly performing prm file.
- thermal and mechanical input files using the using the chosen base prm file. The mesh settings must be:

  *PBPA = 4, the solver will override this if set to any other value

  *PBLR = 0, the solver will override this if set to any other value
- The calibration .json file. An example calibration file can be downloaded from the Autodesk help site.

The PRM calibration file

The prm calibration file has 5 elements:

- Thermal input file - this should point at the full thermal input file name.

- Mechanical input file - this should point at the full mechanical input file name.
- Measurement - Enter in the distortion measured from the experimental build. This equals Nominal width - Caliper measurement. This must be signed, and for most material the measurement will be negative.
- Relative error - Enter in the accuracy of your measurement tool or method.

To execute the calibration process enter the following command.

Windows: `prm_gen /c example.json`
Linux or MacOs: `prm_gen -c example.json`

Note: This process requires 3-5 simulations of a finely meshed part, resulting in run times around 3 hours on a 28 core machine.

2.2 File extensions

2.2.1 Input Files

Analysis Control File

<fname>.in
<fname>: a80: file name prefix
This file is read by the FEA solver and defines the analysis type and other parameters using keywords. Each keyword starts with the * character and contains four more characters. If there are more characters in the same line, they are ignored. Lines starting with # are comments and are ignored. A listing of all keywords is available in the Index 6.3.7.

Batch list File

<fname>.que
<fname>: a80: file name prefix
This file is read by the Netfabb Simulation wrapper when the -q option is used to generate the script <fname>.batch which then submits multiple jobs sequentially in batch mode. This file contains a list of input files (analysis control files w/o .in).

2.2.2 Output Files

Message Output File

<fname>.out or <fname>.log, if the -l option is used during the execution of Netfabb Simulation .

Results Output Files

Files written in . directory:

File name	Result type
<fname>.bin	Binary temperature results file input into mechanical analysis
<fname>.case	Mesh and simulation results for post-processing
<fname>.DDM	Element activation history
<fname>.lsr	Laser path file

Option files written in . directory when using the *WRTU card:

File name	Result type
<fname>.wrtu	Nodal displacement results
<fname>.wrtu.epp	Nodal plastic strain results
<fname>.wrtu.ept	Nodal elastic strain results
<fname>.wrtu.eqp	Nodal equivalent plastic strain results
<fname>.wrtu.sd3	Nodal principal stress results
<fname>.wrtu.sig	Nodal Cauchy stress results
<fname>.wrtu.sp3	Nodal principal stress direction results
<fname>.wrtu.svm	Nodal Von mises stress results
<fname>.wrtu.tmp	Nodal temperature results

Files written in *./results* directory:
These are the results files that can be read by Autodesk Simulation Utility for Netfabb, Paraview, or any other visualization software compatible with the Ensight Gold file format, which are written by default:

File name	Result type
<fname> _bbbbb.geo.ens	Mesh for post-processing
<fname> _bbbbb.tmp.ens	Nodal temperature for post-processing
<fname> _bbbbb.dis.ens	Nodal displacement for post-processing
<fname> _bbbbb.sig.ens	Nodal Cauchy stress for post-processing
<fname> _bbbbb.svm.ens	Von Mises stress for post-processing
<fname> _bbbbb.spr.ens	Principle stress for post-processing
<fname> _bbbbb.mlt.ens	Melt indicator for post-processing
<fname> _bbbbb.case	Mesh and results for post-processing

These are the results files that can be read by Patran by MSC Software, which are written when the optional *ADDP card is enabled in the thermal and mechanical input files:

File name	Result type
<fname>aa_bbbbb.tmp	Nodal temperature for Patran post-processing
<fname>aa_bbbbb.flu	Nodal flux for Patran post-processing
<fname>aa_bbbbb.dis	Nodal displacement for Patran post-processing
<fname>aa_bbbbb.str	Element stress for Patran post-processing
<fname>aa_bbbbb.sig	Nodal stress for Patran post-processing
<fname> _bbbbb.out	Mesh Patran neutral file for adaptive analysis

aa: is the sensitivity variable number. If aa=00, file contains analysis results. bbbbb: is the time increment number.

Chapter 3

Keyword Listing

3.1 Variable Type Notation

Many analysis cards will require the user to enter an input. These variables can be one of the following three types:

- aj : akk : Alphanumeric character input j holding a string of no more than kk characters
- ij : i*k : Integer input j consisting of k bytes
- rj : r*k : Real number input j consisting of k bytes

3.2 System checks

*IOBN: disable Input Output BeNchmark

*IOBN
i1, r1: Disk Check control, Unused variable
*IOBN is used to disable the automatic disk check that occurs at the beginning of each simulation. This disk check warns the user if they are running on a disk type that will slow solver performance. The types of disks that are checked for are USB drives, Network connected drives, and encrypted drives. Zero or positive values for i1 will perform the check, negative values will disable the disk check. The real variable r1 is not used at this time, but must be included.

3.3 Input/Output

*TITL: Header line

*TITL
a1
a1: a80: heading
This option allows a title to be assigned to the analysis. The title will appear in the result file headers.

*INPU: Input neutral file name

*INPU
a1

a1: a80: name of input neutral file
This card specifies the name of the Patran® neutral file (mesh file) to be used in the analysis. **This card can only be used for direct process simulation, not for powder bed processes.** Refinement (but not coarsening) of the neutral file mesh is possible using the *ADAP, *ADP1, *ADP2, *ADP3, or *ADPM cards.
Best Practices: This card is only needed if the user wishes to use a custom user-generated mesh. The card is not needed if the user wishes to auto generate a mesh using Netfabb Simulation (recommended).

*BINA: Binary results output switch

*BINA

This card is used to store temperature history results into a direct access binary file (*.bin). The *.bin file contents are used as an input for the succeeding temperature dependent mechanical analysis. This option is not needed if it is not desired to run a mechanical analysis following the thermal analysis.

*DEPE: Dependent analysis file prefix

*DEPE
a1
a1: a30: dependent analysis file prefix
This option is for mechanical analyses and allows for the thermal results that the analysis depends on to be specified. Temperature history is a required input into the mechanical simulation, meaning that all mechanical analyses must use this option. This requires the dependent thermal analysis to have been completed successfully using the *BINA option.

*STYP: Structure TYPe output for thermal model results

*STYP

This card will enable writing of the structure type results to the thermal results files.

*REST: Restart File

*REST
a1

a1: a80: Restart file path and dependent restart file input name name, without the .in extension. a1 is appended by _r.bin during simulation

This option is used when restarting a part-scale LPBF analysis and allows the user to specify the name of the restart file being used. **This functionality is not enabled for moving source models.** This requires that a previous simulation being run with the *ORES card. The path needs to be included as the solver will access more files than merely the *_r.bin file. Typically this will be the `results` folder.

*OFNS: Output File Number Size

*OFNS
i1

i1: i*4: number of significant figures used in output files. Default 5

This option allows for the number of digits in the results file names to be specified. If the analysis is expected to exceed 99,999 increments, this card should be used and i1 should be increased from the default value. i1 cannot be less than 1 or greater than 10.

*ORES: Output to Restart Files

*ORES
i1

i1: i*4: Output to restart files frequency.

This option allows for a restart file to be written every i1 increments for part scale LPBF analyses. Negative values can be used to write over the same restart file. **This functionality is not enabled for moving source models.** Using positive values will create a series of files of the form `inputname_1_r.bin` in the results directory, counting upwards. Using negative values will create a single file in the results directory of the form `inputfilename.bin`

*OWFC: Output Write File Frequency

*OWFC
i1

i1: i*4: Output write file frequency. Default 1

This option permits the user to choose the frequency at which to write output files. This can be useful in reducing the amount of disk space the result files occupy during large simulations. Output files are written every i1 increments. This option applies to DED models, PRM generation simulations and part-scale mechanical analyses.

*OOPP: Output Only Post Processing increments

*OOPP
*OOPP is used to write only the post process increments of Part Level Powder Bed simulations, for users who want to minimize disk usage and are only interested in post process results.
When used for a Thermal part level simulation, only one increment of results file is written which includes the mesh preview and if available, interlayer temperatures, lack of fusion, hotspot results. Temperatures are not recorded for the build process or for heat treatment analysis, if enabled.
When used in a Mechanical part level simulation, results files are only written for the final post-processing step, part cooldown, heat treatment steps if enabled, support structure removal step if applicable, and build plate removal.

*OFLX: Output heat FluX

*OFLX

This card turns on the output of flux (*.flu) files. These files record heat flux vectors for each node during thermal analyses. The fluxes are recorded to a new file for every increment. These results are also written to the .case files. By default output is set to off.

*ODTX: Output thermal gradients

*ODTX

Using this card will output thermal gradients. This is compatible with both moving source and part level analyses.

*POOL: output meltPOOL volume

*POOL
r1
melt pool temperature
This card will record the melt pool volume at each time step in a CSV formatted .melt file. This card can only be used for moving source models.
Required Cards: *LSRF or *LSRP

*ENSI: ENSIght file output format

*ENSI

This card enables Ensight gold output files. This card is enabled by default.

*ENSA: ENSight Ascii file output format

*ENSA

This card switches file output format from binary to ascii for .case, .geo, and other Ensight files.

*OVTK: Output VTK results

*OVTK

Using this card creates VTK results files in the results folder of the type *.vtu. This only applies to part scale mechanical simulations.

*AVTK: ASCII VTK results

*AVTK

Using this card in conjunction with *OVTK creates ASCII VTK results files.

*ADDP: ADD Patran output format

*ADDP

This card adds Patran outputs in conjunction with the default Ensight outputs. This option is off by default.

*WRTU: Write ResulT Displacements

*WRTU

This card makes the mechanical simulation generates the files <filename>1.wrtu and <filename>2.wrtu with the exterior surface node displacement data before and after release from the substrate, respectively. It will also output temperatures at the end of the simulation at the same increments.
If *FSUB is enabled, <filename>1.wrtu gives the results after release from the the substrate and <filename>2.wrtu records the results after the build is removed from the substrate.
The WRTU file format is X,Y,Z location X,Y,Z displacement, nominally in mm.
*WRTU can also be used be used to export stress, and strain results. These will be exported if the cards enabling their output are included in the input files. Refer to the output type documentation for details on these files 2.2.2.

*ENSF

*ENSF

This card writes results to the finest mesh. This should be enabled when proving moving moving source lack of fusion simulations using *LFUS.

*CAS1

*CAS1

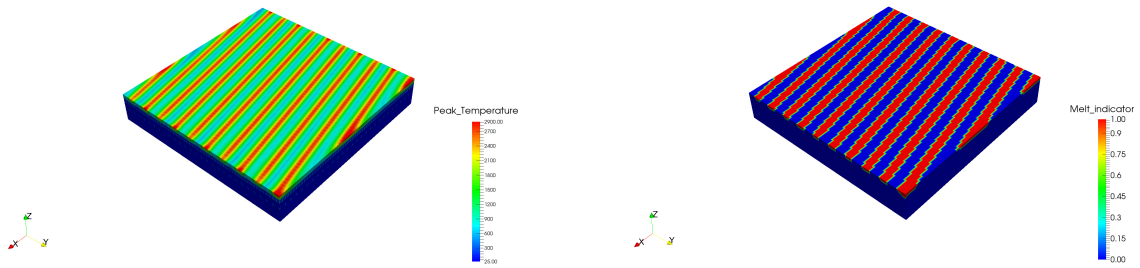
This option only writes a single case file in the working directory, the ./results folder. By default, there is a separate results/*.case file for every increment.

*LFUS: Lack of FUSion

*LFUS
r1

r1: Melt Temperature (or other test temperature)
*LFUS creates two new output result in the Ensight format for moving heat source thermal simulations:
1. Melt Indicator: This Paraview field will produce a binary color contour plot where any element which experiences a temperature greater than r1 has a value of 1 and any temperature less than or equal to r1 has a value of 0. This allows the user to investigate regions that may experience lack of fusion due to the chosen process parameters.
2. Peak temperature: This field records the peak temperature during each time increment.

Examples of these two new fields are shown in Figure 3.1.



(a) Peak temperatures (b) Melt indicator

Figure 3.1: *LFUS results

Best practices Use with *ENSF to improve resolution of *LFUS results. Note: this will create aberrations in the normal temperature contour as temperatures from non active nodes will still be shown.

*TPRE: Temperature PREvious to heating

*TPRE
Optional arguments
r1: Threshold temperature 1
r2: Threshold temperature 2 ...

Using *TPRE records a new output text file, filename$$_tpre.txt, where filename$$ is the input thermal file *.in name, without the extension. This file records the time increment, the X,Y,Z coordinates of the laser heat source at the current time increment, the starting temperature prior to heating, and a binary flag for the preheat temperature(s) which report a value of 1 for any increment temperature that is above the temperature of interest, and 0 for any temperature less than or equal to the temperature of interest.

*INTA: write all peak temperature increments

*INTA

This card ensures that peak temperatures are written at all increments instead of just the last increment, which occurs by default.

*STAT: orientation optimization STATistics export

*STAT

i1

When executing an orientation optimization process use this card to record mechanical results to write additional details to the simsignal file, which is then recorded in the final results.csv file

output by the optimizer. The variable i1 is used to control the number of points used to calculate the median values.

3.3.1 Part Level Stress Output

The following options enable the user to output various stress outputs during part-level powder bed simulations. Note that at this time, due to the approximations used by the solver, **these stresses are qualitative unless *PPLA is enabled**. These can still be used to investigate build failure and residual stresses, but should not be taken as quantitative estimates of stress.

*OSIG: Output Cauchy stress tensor values

*OSIG Enables Cauchy stress output. This is a tensor output. Use *PPLA to enable quantitative residual stresses.

*OSD3: Output Principal stress direction

*OSD3 Enables Principal stress direction output. This is a vector output.

*OSP3: Output Principal stress values

*OSP3 Enables Principal stress values output. This is a vector output. Use *PPLA to enable quantitative residual stresses.

*OSVM: Output Von Mises stress values

*OSVM Enables Von Mises stress output. This is a scalar output. Use *PPLA to enable quantitative residual stresses.

*PPLA: Post cooling PLAsticity

*PPLA

This option turns on quantitative stress computation after the cool down step, which results in the solver performing several nonlinear analysis steps at the end of the standard mechanical simulation cycle. This ensures higher accuracy of stress results at the expense of added solution time. Stresses are quantitative post-cool down, during bolt removal via *FSUB, heat treatment simulation, support structure removal, and post process removal from the build plate. If the build plate material is of a different material assigned by *PBS2 or *PBSN, ensure that a set of constant temperature plasticity properties are assigned to the build plate material block using the *PLAS card. Note that when using this card, stress results during processing will not be recorded.
Best practices: Use *PPLA whenever residual stresses are going to be analyzed or validated. Also use *PPLA whenever validation or analysis will be performed after build plate release (post *FSUB steps) or post build plate removal, as the excessive residual stresses used without *PPLA will cause excessive warping of these components. *PPLA is also required to perform a heat treatment simulation.
When using *PPLA, additional care may be necessary to ensure the mesh has converged. Notably it is advised that lower *PBLR values be used, 0 or 1, as higher values have exhibited non-converged, non-physical results.

It is advised that when using *PPLA, the *PBSS card not be used, as this adds additional constraints to the model which may lead to poor simulation results.

*MPLI: Minimum Newton-Raphson PLastcity Iterations

*MPLI
i1,r1
i1: Minimum number of plasticity iterations. Default 1
r1: Minimum plasticity step. Default 1d-15

By default the Newton-Raphson solver assumes the non-convergence whenever the residual between two successive *PPLA plasticity increments is greater or equal to 10%. Use *MPLI to force the solver to continue iterating up to at least i1 iterations. Use r1 to set the threshold for the minimum sized time step the plasticity algorithm will be allowed to attempt.
Required Cards: *PPLA

3.4 Analysis control

*ANTP: Analysis Type

*ANTP
i1

i1: i*4: Analysis type number

2: Transient Heat Transfer
4: Quasi Static Mechanical

This card is used to specify the analysis type.

*AXSP: AuXSPar array size scaling factor

*AXSP
r1, r2
r1: r*8: Scaling factor for initial condensation array vector. Default 1
r2: r*8: Growth scaling factor for condensation array, Default 1.

This card is used to increase the maximum size of the condensation solution array. Adjusting r1 will increase the initial size of the matrix array. If this array is too small, the growth scaling factor multiplies the initial array size by 1.10*r2. This process is repeated until the array is large enough to contain the entirety of the condensation array.
This option should be used when the program exits during the auxspar phase of the program initialization, for extremely large or extremely fine meshes.
Best Practices: Use *AXSP only for simulations which do not complete the meshing step with the default parameters. Good initial values would be r1 = 1.1, r2 = 1.1.

*CMSH: Check MeSH

*CMSH
This card pauses the simulation after the auto-generation of a mesh during the thermal analysis to let the user preview the mesh. It will prompt the user to continue the simulation or exit. This allows the user to examine the auto-meshing of a new geometry without running the full simulation. This card has no effect during the mechanical portion of the simulation.

*END: END input file

*END
This card terminates reading from the thermal and mechanical input files. *END is required at the end of every thermal and mechanical input file.

*NLTL: Total Lagrangian analysis switch

*NLTL

This option allows for mechanical analyses to be performed using the Large Deformation formulation. By default, the small deformation formulation is used.
This feature can be used to anticipate buckling. If buckling occurs, it causes a non-convergence in the solver. If the *NLTL card is enabled and the simulation fails to converge then an additional error message is shown in the log after failure:
`Error: Solver failed to converge using Large Deformation Formulation. Buckling may have occurred.`

*NOFC: NO automatic OFf core pre-processing

*NOFC

By default the solver will make a memory estimate at the beginning of the simulation process to determine if the available RAM is sufficient to solve the problem. If the RAM is not sufficient, the solver will enable *OFC2 and *OFC3 to write matricies to Direct Access Files. To override this and to directly turn on or off the off core preprocessing cards *OFC1, *OFC2, and *OFC3, the user must include the *NOFC card.

*OFC1: On Core pre-processing stage 1

*OFC1

This option writes certain pre-processing arrays to RAM. This option can reduce computational time, but will do so at the expense of system resources.
Best Practices: This option should be used only for small geometries, or for systems with more than 120 GB of RAM. If the 'Insufficient Virtual Memory' error message is encountered, remove *OFC1 from the input files.
Required Cards: *NOFC.

*OFC2: OFf Core pre-processing stage 2

*OFC2

This option writes certain pre-processing arrays to Direct Access files instead of allocating in RAM. Enable this option to reduce the RAM requirements demanded by simulations using large numbers of degrees of freedom. This option can be used in conjunction with *OFC3 for even more memory intensive simulations.
Required Cards: *NOFC.

*OFC3: OFf Core pre-processing stage 3

*OFC3

This option writes certain pre-processing arrays to Direct Access files instead of allocating in RAM. Enable this option to reduce the RAM requirements demanded by simulations using large numbers of degrees of freedom. This option can be used in conjunction with *OFC2 and *OFC3 for even more memory intensive simulations.
Required Cards: *NOFC.

*MAXM: MAXiuM allowable RAM usage

*MAXM
r1

Use *MAXM to automatically stop a simulation if the amount of RAM used by the system is greater than r1%. Allowable values of r1 are 0-100.

*KTMP: Keep TeMPorary files

*KTMP

This option keeps the temporary out of core files produced by using *OFC2 and *OFC3. By default these files are deleted.

*RELA: RELAxation Control

*RELA
i1, r1

i1: i*4: number of iterations. Default 0
r1: r*8: Scaling factor. Default 1.d00

This card allows for the step size of the Newton-Raphson solver to be scaled by r1 for il iterations. The purpose of this option is to stabilize the solver and/or accelerate convergence.
Best Practices:
Thermal analysis should converge using 1, 0.4 Excessive non-linearities caused by material properties or using *LATE may require using more relaxation iterations, i.e. 2, 0.4.
Mechanical analysis should converge 1, 0.2 This is the default value used by GUI. If the simulation does not converge increase the number of iterations, i.e. 2, 0.2

*RELM: RELAxation Mechanical values

*RELM
i1,i2,r1
i1: 1 If mechanical numerical relaxation is used, 0 (or other value) otherwise.
i2: Number of mechanical relaxation iterations.
r1: Mechanical relaxation scaling factor.

*RELM is used to ensure the relaxation values from both the thermal and mechanical PRM generation simulations are written to the resulting PRM file. This allows Simulation Utility for Netfabb to read and display this value in the Processing Parameters library, which allows users to create new PRM files rapidly from existing PRMs, which if *RELM is enabled, will inherit the relaxation values specified by the user.
Required Cards: *RELA, *GTAB

*SOLU: SOLUtion Parameters

*SOLU
i1, r1, r2

i1: i*4: max number of iterations. Default 30
r1: r*8: residual tolerance. Default 1.d-2
r2: r*8: max residual. Default 1.d20

This card controls the Newton-Raphson solver. If the solution does not converge within i1 iterations at any given time step, the time step size will be reduced and the solver will cutback. Once the residual is smaller than r1 the simulation will continue to the next time step. If the max residual exceeds r2, the analysis will be automatically terminated.

*SPA8: SParse Array 8-byte

*SPA8
*SPA8 forces the solver to use 8-byte integer arrays for sparse data structures. By default the solver automatically switches from 4-byte to 8-byte when the 4-byte limit is exceeded.

*TRAN: Transient analysis timing control

*TRAN
r1, r2, r3, r4, r5, r6, i1, i2

r1: r*8: start time
r2: r*8: end time
r3: r*8: initial time increment
r4: r*8: maximum allowable time increment
r5: r*8: minimum allowable time increment
r6: r*8: incrementation tolerance
i1: i*4: maximum number of cutbacks
i2: i*4: maximum number of increments

- This card sets the simulation start and end time as well as the initial, maximum, and minimum time increments.
- The card is also used to specify the incrementation tolerance which controls how quickly time steps increase to the maximum value during cooling.
- Also sets the maximum number of allowable cutbacks and increments. If the maximum number of cutbacks or increments is exceeded, the analysis will be terminated.

For DED simulations the *TRAN card is optional. These options and *TAUT will be set according to best practices and the values specified by the *LSRF card. This will add an hour of cooling time after the build simulation has completed to account for post process cool down.
Best Practices:

r3: To avoid aliasing, analysis of direct processes and moving source models should have increments so that the time steps are equal to the time it takes for the laser to move 1 laser radius. This can be automated by using *TAUT to control the time incrementation.
r4: Allowing for larger maximum time increments (10-1000s) can decrease the time it takes to complete a simulation for those builds with larger dwell times or long post-process cool down times. However if the time steps are large compared to the time scale of the cool down periods, there may be an undesirable loss of resolution.
i2: If the simulation will take more than 99,999 increments, ensure *OFNS is used so the simulation does not end prematurely.

*TAUT: Automatic Time Incrementation Based on Heat Source Radius

*TAUT
r1
r1: r*8: time in sec. Default 1.d0
This option is used in moving heat source analyses to automate time incrementation. When a heat source is active, the time increment is set to radius/velocity*r1, meaning that if r1 is set to 1, the heat source will move a distance equal to its radius at each time step. Using a r1 value 0.5 will move one half laser radius per time step, and so forth.
Best Practices:
For moving source models *TAUT should be enabled and alteration of time steps should be completed using the *TAUT r1 value.

*TPRM: dwell Time multiplier for PRM generation

*TPRM
r1
r1: r*8: time in sec. Default 20
This card is used to alter the timing during PRM generation. *TPRM is used to account for the time required to deposit the rest of the volume of the part or parts that will be simulated using the PRM file which will be generated. The value r1 is multiplied by the recoater time to create the dwell between the layers simulated by the moving source PRM generation model.

*ZDWL: Z height DWelL time

*ZDWL
r11, r12
r21, r22
...

The inclusion of the *ZDWL card into a thermal part scale analysis will add the time r11, in seconds, at the height r12, in mm. This can improve model accuracy for builds that experienced either a planned or unplanned stoppage, allowing for accurate models of builds that run out of powder, have minor recoater interference issues, or are paused at the end of the work day.
Dwell times have to be positive, any negative times will be rounded to 0. Any dwell time added to Z heights lower than the base of the build component will be added prior to the first layer. Any dwell times for heights above the part will not be included. Dwell times at heights that are exactly at that layer group height will be added during that layer group has simulation. For dwell times

in between two layer groups, the time will be rounded up and added before the next layer group is simulated.

*TWAL: Wall CPU limit

*TWAL
r1
r1: r*8: time in sec
This option terminates execution and saves results if the wall CPU time exceeds r1. This option is useful when running in clusters using PBS so that the results are saved before the analysis is terminated.

3.5 Element Activation Strategies

Elements used in the analysis can exist in the following three states: [1]:

- Active Elements: These elements are included in the analysis and given their true material properties.
- Quiet Elements: These elements are included in the analysis but are given material properties such that they do not effect the simulation. For thermal simulations the thermal conductivity and specific heat are scaled down. For mechanical analyses the elastic modulus is scaled down.
- Inactive Elements: These elements are not included in the analysis.

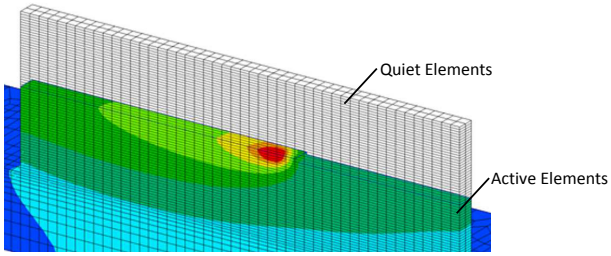


Figure 3.2: Illustration of element activation

*DDM!: Automated activation by layer using hybrid inactive/quiet method

*DDM!
r1, r2, [r3]

r1: r*8: top z coordinate of substrate
r2: r*8: bottom z coordinate of substrate
r3: r*8: time offset to activate layers, in seconds. Default radius/velocity/4

This DDM option switches elements from inactive to quiet on a layer by layer basis. The time of layer activation is set to the start of deposition of each layer minus r3. Then, Netfabb Simulation searches which elements belong to each layer based on their z coordinate. The elements are activated when contacted by the heat source.
For auto generated meshes this card also defines the substrate or build plate thickness. r1 should correspond to the bottom coordinate of the .stl file or the heat source path and r2 defines the thickness of the auto generated substrate.

*DDM1: Parameter definition for DDM methods

*DDM1
r1, r2, r3, [r4]

r1: r*8: scaling factor for thermal conductivity. Default 1.d-6
r2: r*8: scaling factor for specific heat. Default 1.d-2
r3: r*8: scaling factor for elastic modulus. Default 1.d-4

r4: r*8: scaling factor for emissivity. Default 1.d0

This option allows the definition of the scaling factors used for quiet elements and powder elements when using the *+PDR card. The default values are for quiet elements. The suggested values for powder elements are:
r1: r*8: scaling factor for thermal conductivity = 0.01
r2: r*8: scaling factor for specific heat = 1
r3: r*8: scaling factor for elastic modulus = 1.d-4
r4: r*8: scaling factor for emissivity = 1.0

Required Cards: *DDM!

*DDMC: Option to define Clamp in conjunction with *DDM!

*DDMC
i1
i1: i*4: Configuration ID of build elements

This DDM option is used in conjunction with *DDM! to differentiate the build areas of the mesh above or below the substrate from fixturing and clamps.
This DDM option can also be used in conjunction with *ADAP to define areas of the base mesh that do not need to be processed for adaptivity. These elements will not be refined.
Required Cards: *DDM!, *ADAP

*DDMM: Option to define material ID for quiet or powder elements

*DDMM
i1
i1: i*4: Material ID of quiet or powder elements

This option allows for use of a material ID to define the material properties of quiet or powder elements. When used, quiet elements are assigned this material ID initially. After the elements are switched to active, the element material ID is switched to that of the solid material. When *DDMM is used, no scaling of properties is used for quiet elements (Values defined by *DDM1 are ignored).
Currently implemented for thermal analysis only.
Required Cards: *DDM!

*DDMP: Switch for powder-bed modeling during moving source modeling

*DDMP
The option should be used when powder elements are present and has the following effects:

- The temperature of the quiet elements is not reset to the initial temperature upon their activation.
- Layers contain all elements, even those not activated as the powder is always included in the analysis, unlike direct processes, where the material is added.

Required Cards: Use *DDM1 or *DDMM to define powder properties and *DDM! to define element activation.

*DDMZ: Z direction element floating

*DDMZ
This option allows elements to float in the z direction during the mechanical simulation. This enables the modeling of direct processes (e.g. Electron Beam or Laser Direct Energy Deposition) using the multi-scale modeling approach.

3.5.1 6 Axis DED simulations

*DDM6: 6 Axis DED - Tech Preview

*DDM6
This card enables 6 Axis DED type simulations. This card in enabled automatically whenever *LSR3 is used.

*LSR3: 6 axis LaSeR file - Tech Preview

*LSR3
a1 : 6 Axis type LSR File name

6 Axis DED simulations are controlled by the laser path specified by the *LSR3 card. The file format is nearly identical to the traditional 2.5D *LSRF file but with one additional column used to specify the layer group number. The file format is:
r11, r12, r13, r14, r15, r16, r17, r18, r19, r1(10), r1(11), r1(12), r1(13), i11
r21, r22, r23, r24, r25, r26, r27, r28, r29, r2(10), r2(11), r2(12), r2(13), i21
...
r11: r*8: power
r12, r13, r14: r*8: vector of heat source direction
r15, r16, r17: r*8: start point
r18, r19, r1(10): r*8: end point
r1(11): r*8: melt pool radius
r1(12): r*8: velocity
r1(13): r*8: start time
r1(14): i*1: layer group number
...
The layer group number controls when elements are specified as quiet elements. For elements in layer groups that have not yet been reached, they remain 'dead' elements, not part of the solution matrices. Once that layer group is activated they become quiet, using the scaling factors set in *DDM1. Once that layer group is finished and the next is begun, any elements not activated by the laser heat source are made 'dead' elements again and removed permanently from the solution matrices. Layer groups must be in increasing order, but have no other requirements.
6 Axis simulations require a substrate STL as specified by *STLM.
Best Practices: Specifying the number of layer groups requires a balance between the desire to have as few quiet element activated at a time in order to minimize the computational time it takes to resolve the problem, while also not switching layer groups so often that the renumbering process that occurs during each activation step slows down the simulation speed excessively. Typically a

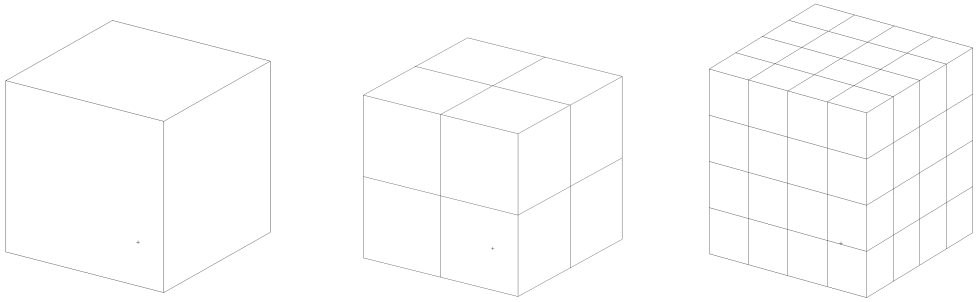
single uninterrupted laser pass is good starting place for layer group division. However for very large parts or for deposition process with no dwells, additional layer grouping may be necessary.
Required Cards: *STLF, *STLM

Adaptive meshing controls

*ADAP: ADAPtivity generations

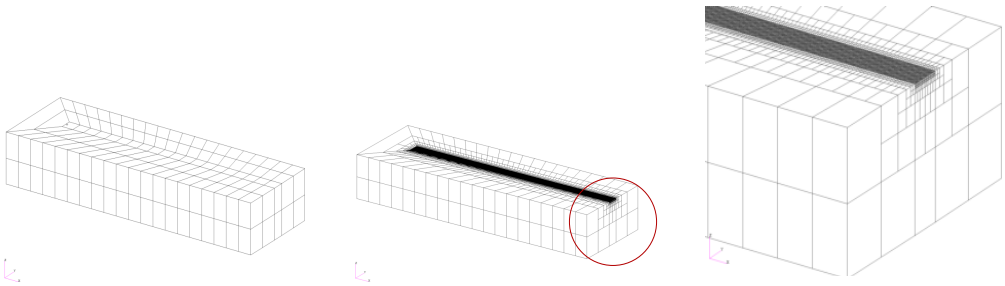
*ADAP
i1
i1: i*4: Levels of element refinement. No default value.

This option specifies the number of levels of element refinement for adaptive analyses in all three spacial dimensions. The maximum value of i1 allowed is 6. Levels of refinement are demonstrated below:



(a) No refinement (b) 1 level of refinement (c) 2 levels of refinement

Figure 3.3: Hex8 element subject to various levels of refinement.



(a) Unrefined mesh (b) Refined mesh with 2 levels of refinement (c) Magnified refined mesh

Figure 3.4: Mesh subject to refinement.

Best Practices:
Direct Energy Deposition or Moving Heat Source Powder Bed Simulations:
Using a Netfabb Simulation generated mesh (using *AUTM):

The automatic mesh generation element size is based upon the laser radius size and *NELR (which has a default value of 1). Ensure there are 1-2 elements per laser radius in the laser path and at

least 2 elements through the thickness of the substrate.

*ADAP values of 1-4 are typical.

Using a Patran® generated mesh (using *INPU):

The Patran® generated mesh will be the coarsest possible mesh as the adaptivity routines of Netfabb Simulation will refine but not coarsen the mesh. Produce a mesh with 1-2 elements per laser radius in the laser path and at least 2 elements through the thickness of the substrate, using a combination of Patran® , *ADAP, and *SUB2. Many, but not all, simulations will converge using 1 element per laser radius. Refinement beyond 2 elements per laser radius will prolong simulation times without giving more accurate thermo-mechanical simulations.

Part Scale Powder Bed Simulations

Part scale meshes are generated from .STL files. Meshing of the manufactured component is controlled by *PBPA and *PBLR, while *ADAP controls the coarsening of the substrate. Higher levels of adaptivity yield coarser build plate meshes. A value of 4-5 is adequate for most simulations.

*ADPM: Switch for moving element coarsening/refining within layer

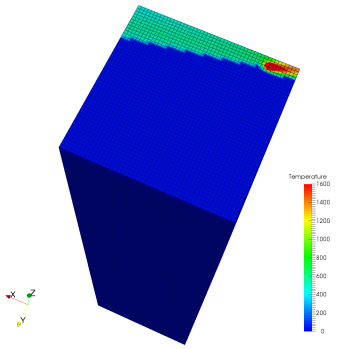
*ADPM
r1
r1: r*8: Tolerance for element coarsening. Default 50.d0

This option turns on moving-source adaptivity and can only be used for moving heat source powder bed *thermal* analyses of a single layer. When the temperature gradient across an element equals r1 or below, the element will be eligible for coarsening.

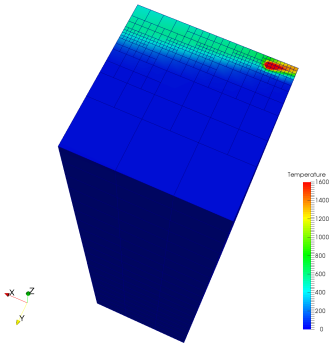
Required Cards: This card requires the use of *DDMP and *LSRP.

The effect of using *ADPM is shown in Figure 3.5 along with the associated single CPU run times. Without using *ADPM the mesh has excessive resolution to resolve the problem. Using the default value of 50 reduces simulation time by 82%. Changing the value of *ADPM to 100 generates a further reduction of run time of 31% over the default value. A similar reduction of time is achieved by using *ADPM of 1000, but note in Figure 3.5(d) thermal abberations occur. These are due to the improper integration of heat over the part.

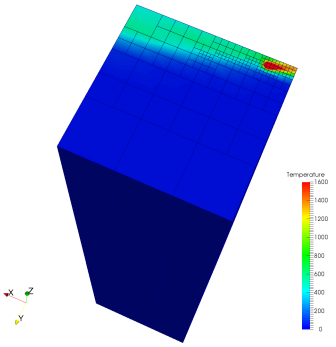
Best Practices
The default value of 50 is a good compromise between speed and accuracy. *ADPM should not be set below 25.



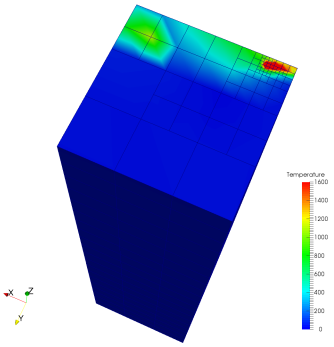
(a) *ADPM = off, Run time = 3797 s



(b) *ADPM = 50, Run time = 665 s



(c) *ADPM = 100, Run time = 461 s



(d) *ADPM = 1000, Run time = 306 s

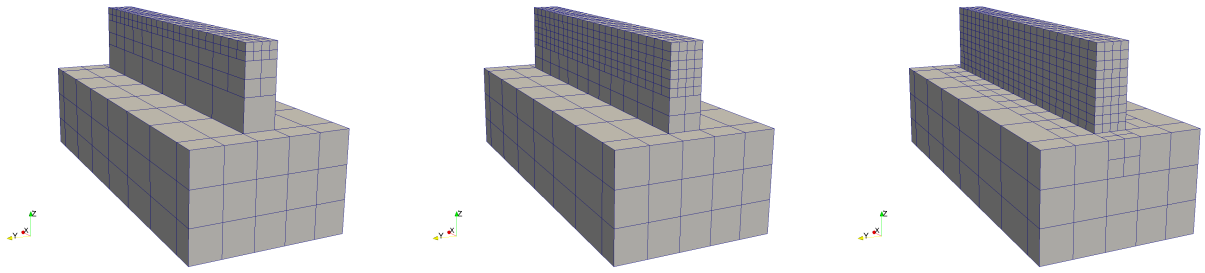
Figure 3.5: Effect of using *ADPM

*ADP1: Adaptive Refinement Control Parameters

*ADP1
i1
i1: i*4: Number of layers, minimum 2. Default 2.

This option controls the number of fine layers beneath the deposition layer when coarsening. The effect of this card is shown in Figure 3.6, using values of 2, 5, and 10, including the single CPU runtime for each simulation.
Required Cards: *ADAP

Best Practices:
*ADP1 is used to refine layers below the deposition to preserve simulation information and to prevent over coarsening of the mesh, which can result in an artificially high stiffness, diminishing the accuracy of the mechanical simulation. At a minimum there should be 2 elements through the thickness of the substrate throughout the simulation history, which may be achieved by using either *ADP1 or *SUB2.



(a) *ADP1 = 2, Run time = 27 s (b) *ADP1 = 5, Run time = 42 s (c) *ADP1 = 10, Run time = 66 s

Figure 3.6: Effect of using *ADP1 on the mesh and single CPU run time

*ADP2: Adaptive Relaxation Control Parameters

*ADP2
i1
i1: i*4: Number of relaxation iterations after refinement/coarsening in mechanical analysis, minimum 2. Default 5. Increase this number above 5 to aid mechanical simulations which fail to converge at increments when a new layer of elements is activated.

Required Cards: *ADAP

*ADP3: Adaptive option to define element to be refined

*ADP3
i1
i1: i*4: Configuration ID of build elements

This option is used in conjunction with *ADAP to define elements that will be refined. Elements with different configuration ID will not be refined. This can only be used in conjunction with a Patran generated mesh, using *INPU.
Required Cards: *ADAP, *INPU.

*SUB2: SUBstrate force 2 element per thickness

*SUB2

This card is used in combination with *ADAP to instruct the adaptivity algorithm to use at least 2 elements in the thickness direction of the substrate.
Required Cards: *ADAP

*SYMM: SYMMetry plane definition for adaptive analysis

*SYMM
i1, r1
i2, r2

...

i1: i*4: normal to plane axis 1 for x, 2 for y, 3 for z
r1: r*8: coordinate of plane location
...

This option allows for the definition of symmetry planes in either moving source DED or part scale LPBF analyses. For LPBF simulations, the stl file must already have been sectioned prior to importing for use in conjunction with the *SYMM card. The symmetry plane specified must align with the substrate. To override this requirement use *SYM2.
Required Cards: *ADAP

*SYM2: SYMmetry for non-aligned planes

*SYM2

Symmetry using the *SYMM card requires that the plane used is aligned with the substrate. Using *SYM2 overrides this requirement.

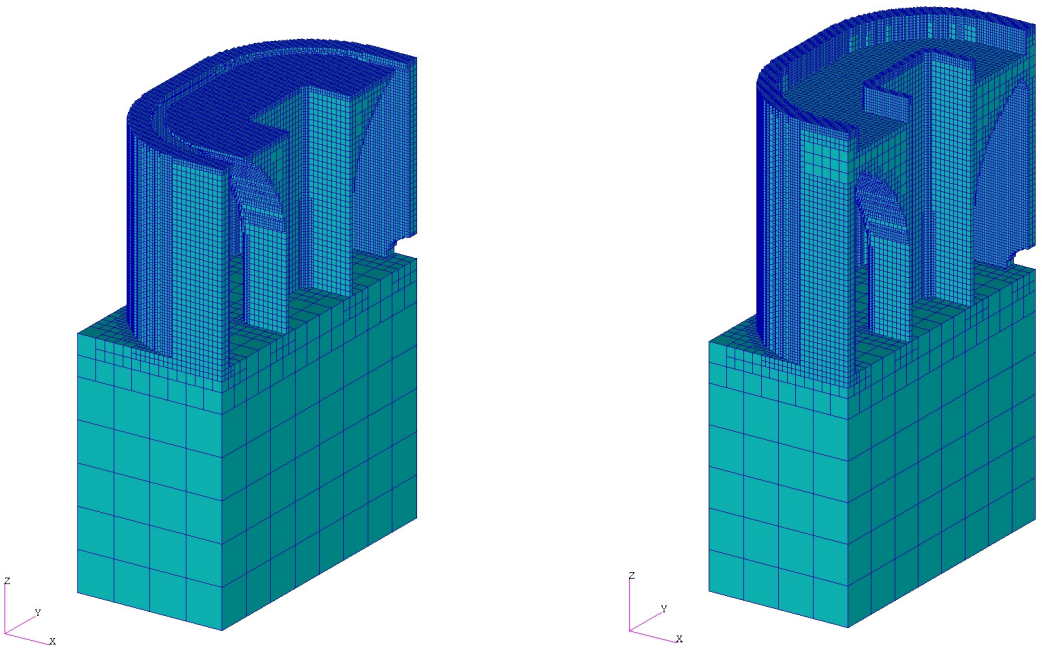
*NBBX: No Bounding BoX change

*NBBX

In some cases when STL repair is engaged, the bounding box may be slightly changed. This can interfere with the *SYMM card or mechanical constraints such as *FIXC and *FIXR, as the origin is moved. By default, after pre-processing is completed, the bounding box is reset to initial value so that cards which contain user specified coordinates align with the coordinate system used in the FE mesh. Use *NBBX to override the bounding box reset, using the updated bounding box after part repair.

3.6 Part-Level Powder-Bed Modeling

Part-Level Powder-Bed modeling involves activating elements in groups of layers. The feature is triggered by the *PBPA card and requires adaptivity (*ADAP), substrate definition (*DDM!), and Process Parameter File (*PBPF) cards. Each layer group is activated at a fine element scale, as additional layers are deposited on the top, elements below are coarsened. Figure 3.7 shows the mesh during activation of the 72nd and 91st layer groups.



(a) Mesh after activation of 72nd layer group

(b) Mesh after activation of 91st layer group

Figure 3.7: Part-Level Powder-Bed Modeling.

Part-Level Powder-Bed modeling involves both a thermal and a mechanical analysis. Currently, the powder is removed from the mesh and heat loss into the powder is modeled as convection *CONV. The mechanical analysis uses the temperatures computed from the thermal analysis to compute the mechanical response. The process parameters for part scale modeling are stored in a separate file and read using *PBPF.

In Part-Level Powder-Bed modeling, the input mesh can be automatically generated by importing an STL file using *STLF. When the *STLF option is used, information entered in the *DDM! card is used to determine the thickness of the buildplate.

Substrate preheating for *PBPA analyses is modeled by using the *INIT card in the thermal analysis only. Continuous controlled heating of the substrate is modeled by the *PBSH and *INIT cards in the thermal analysis only. When substrate preheating or continuous heating is modeled, *INIT in the mechanical analyses should be set to room temperature. The *PBSS card can be used in the mechanical analysis to prevent the substrate from bulging. The *PBIS card can be used to insulate the sides of the substrate during thermal analyses, essentially simulating a build plate with many different builds. By default the substrate size is small, only extending to the bounds of the

.stl box. However, if the user wishes to use a larger substrate the dimensions can be extended using *SBDM or *SBXY. Circular or rectangular fixtures can be added to the substrate using *FIXC and *FIXR, respectively.

*PBPA: Powder-Bed Part-level Analysis

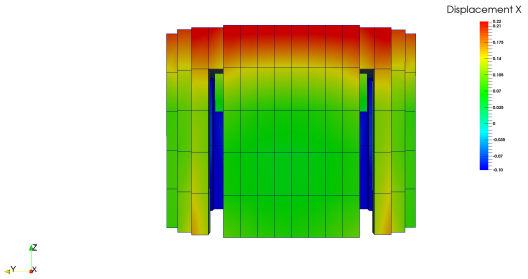
*PBPA
il
i1: i*4: number of layers grouped per element.

This card controls the number of layers combined to form the smallest element thickness. If the deposited layers in the source PRM simulation have a thickness t then the smallest elements will have sides il*t in length. The recommended value then will be:
PBPA = thinnest wall / layer thickness.
By default, the minimum *PBPA value allowed is 5. This may be altered using *MNLR.

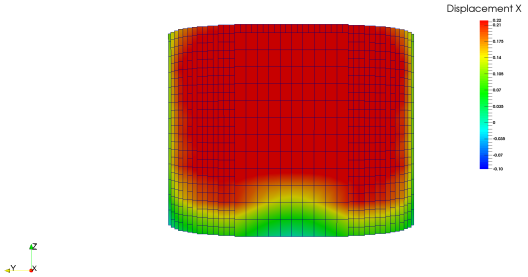
Best Practices: il*t should never be larger than half of the width of the thinnest wall in the structure being modeled. However, this is not always feasible. Using the GUI to generate input files will perform these calculations automatically. Using larger values of *PBPA will dramatically increase the speed of simulations, so the largest value possible should always be used. Good engineering practice dictates a 3 part mesh convergence study with respect to displacements should always be performed by the user for each new geometry.

In Figure 3.8 the effect of using various *PBPA values is illustrated.

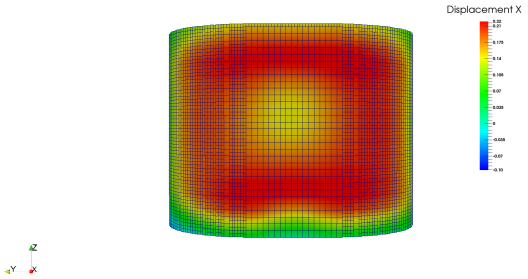
One can see in Figure 3.8(a) that using *PBPA values that are too high will not preserve the geometry of the reference stl file. As the *PBPA values are decreased geometry (Figure 3.8(b)) is preserved, but the mesh may be too coarse to resolve the solution accurately. As *PBPA moves towards the default limiting value of 5, the solution converges. However, reducing *PBPA comes at the cost of increasing run times. Looking at the converged solution for the *PBPA = 10 and 5, the single core run time increases by 6 times.



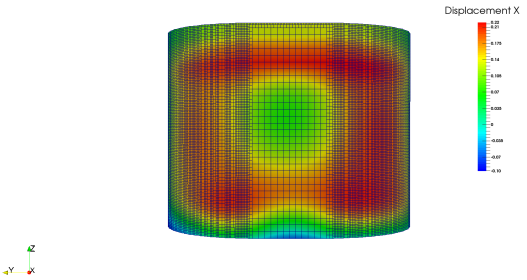
(a) *PBPA=125, Thermo-mechanical run time = 3.12 s



(b) *PBPA=25, Thermo-mechanical run time = 43.9 s



(c) *PBPA=10, Thermo-mechanical run time = 43.9 s



(d) *PBPA=5

Figure 3.8: The effect of *PBPA on the mesh, displacement results, and single core run time.

*MNLR: MiNimum LayeR grouping

*MNLR

i1: i*4: number of layers grouped per element.
This card specifies the minimal allowable number of layers that may be grouped together. By default *MNLR = 5. Use the *MNLR card to override this default value.
Best Practices:
All but the finest features can be accurately meshed using the default values. Using *PBPA values lower than 5 can incur excessive runtimes. Using values lower than 3 is inadvisable.

*PBPF: Powder-Bed Parameter File

*PBPF
a1
a2
...
a1: a30: Process Parameter file name
All Part-Level Powder-Bed analyses require a process parameter file as an input. This card allows the user to select the desired file.
Multiple PRM files may be input when using multiple STL files, which are input using *STLF and assigned PRM files using *STLM.

*+PDR: include PowDeR elements

*+PDR

This card introduces powder elements into the part scale powder bed modeling. This must be used in both the thermal and mechanical analysis files. This allows for more accurate predictions of heat transfer during powder bed simulations. By default powder material properties are scaled so that the thermal conductivity is 0.01 × solid part conductivity, and powder specific heat is 0.60× solid part specific heat. Scaling can be controlled using *DDM1 or the powder can be assigned its own material properties directly using *DDMM in conjunction with the material property card, *MATI. When using this card the powder elements will be visible in the thermal analysis. This is of particular interest when modeling parts that trap powder within their body, when modeling multiple parts on a build plate which are close enough to transfer heat amongst the various parts, or a part with disparate sections which may transfer heat back to itself through the powder. Note that the powder elements will not be displayed in the mechanical simulation, as the powder elements are not attached to the body, and will have no effect upon the mechanical behavior.

*PBIS: Powder-Bed Insulated Substrate

*PBIS

This card turns on insulating sides and bottom of the buildplate in *PBPA analyses. Figure 3.9 illustrates the condition.

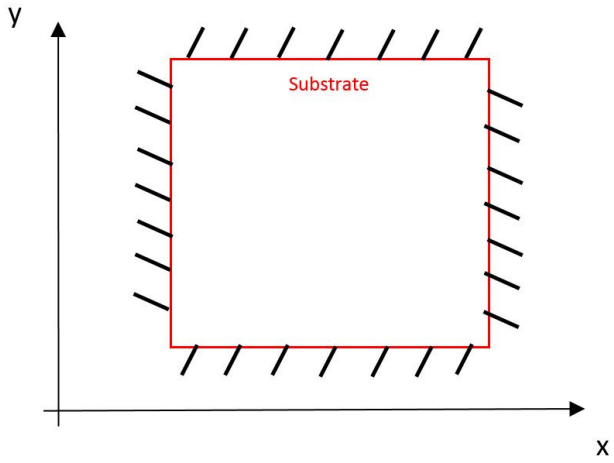


Figure 3.9: Surfaces insulated by *PBIS

*PBSH: Powder-Bed Controlled Substrate Heating

*PBSH
r1
r1: r*8: Substrate temperature

This card fixes the bottom side of the substrate to a temperature r1 in *PBPA analyses. This option represents the condition when controlled heating is used to maintain the buildplate at a desired temperature.

*PBS2: Powder-Bed Substrate material ID 2

*PBS2

This card assigns and uses material ID 2 for the buildplate, allowing the user to make the buildplate a different material than the deposited metal. If *PBS2 is not used, both the buildplate and the part are assigned material ID 1.

*PBSN: Powder-Bed Substrate material ID N

*PBSN
i1

This card assigns and uses material ID i1 for the buildplate, allowing the user to make the buildplate a different material than the deposited metal. If *PBSN is not used, both the buildplate and the part are assigned material ID 1. If both *PBSN and *PBS2 are used, *PBS2 is ignored.

*PBSS: Powder-Bed Symmetry BC's on Sides of Substrate

*PBSS

Figure 3.10 illustrates the effect of this option. This card turns on symmetry boundary conditions on the sides of the substrate when *STLF is used in *PBPA analyses. This card also turns on

symmetry boundary conditions (displacement and heat flux) on the sides of the part when *LSRP is used in *AUTM analyses.

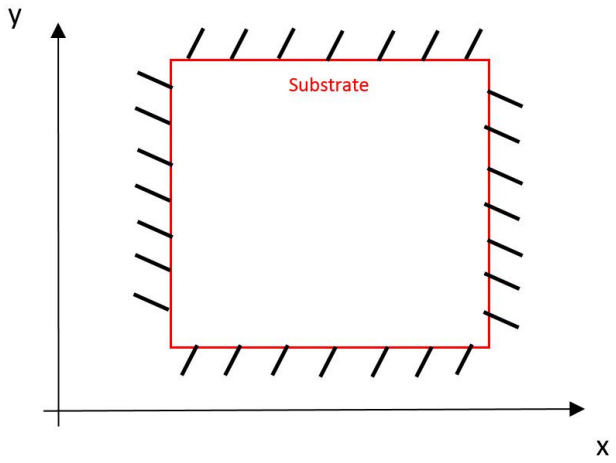


Figure 3.10: Surfaces constrained by *PBSS

Best Practices: It is advised that *PBSS not be used in conjunction with *PPLA which enables part level plasticity, that could result in overconstraining the part leading to non-physical results.

*PBSX: Powder-Bed Symmetry BC's on X sides of substrate

*PBSX

This card also turns on symmetry boundary conditions (displacement and heat flux) on the X axis sides of the part when *LSRP is used in *AUTM analyses. This option represents deposition of finite thickness sections. The option is illustrated in Figure 3.11.

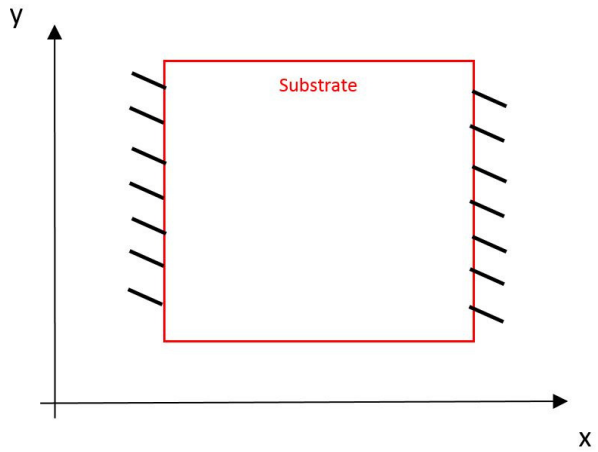


Figure 3.11: BCs assigned by *PBSX

*PBSY: Powder-Bed Symmetry BC's on Y sides of substrate

*PBSY

This card also turns on symmetry boundary conditions on the Y axis sides of the part when *LSRP is used in *AUTM analyses. This option represents deposition of finite thickness sections. The option is illustrated in Figure 3.12.

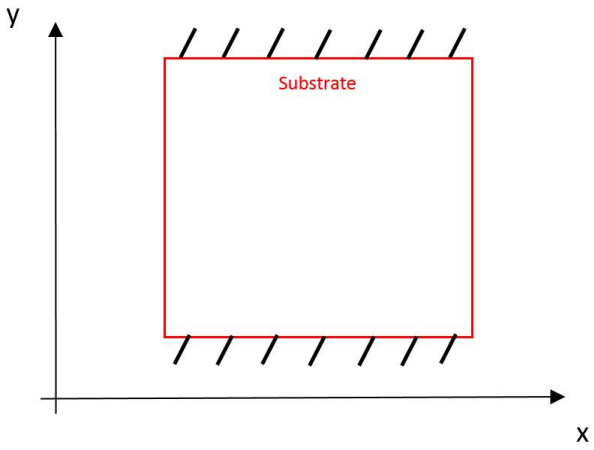


Figure 3.12: BCs assigned by *PBSY

*FSUB: Floating SUBstrate

*FSUB

This card applies displacement boundary conditions on the substrate of part scale powder-bed analyses to simulate rigid fixturing during deposition and release from the machine during cool down.
Figure 3.13 depicts the use of the *FSUB card. During deposition only the z coordinate at the bottom side of the substrate is fixed and the x and are free, except for a circular region at the center of substrate with a radius equal to the largest element size. During the cooling time increment the z coordinate is released to simulate removal of the substrate from the machine. When the *FSUB card is used, the *PBSS card is ignored.
Best Practices: *FSUB is most accurate when simulating an entire build plate, as if just a small part and the build plate directly below is being modeled, the applied boundary conditions are not truly representative of the physical process of bolt removal.
When using *FSUB, part scale plasticity should be enabled to ensure

*SSSS: Switch Support Structure Separation order

*SSSS

By default, after the process simulation and any heat treatment modeling has completed, the support structures and the component are first removed from the build plate together, then the final increment removes the support structures, leaving just the component. This card switches the

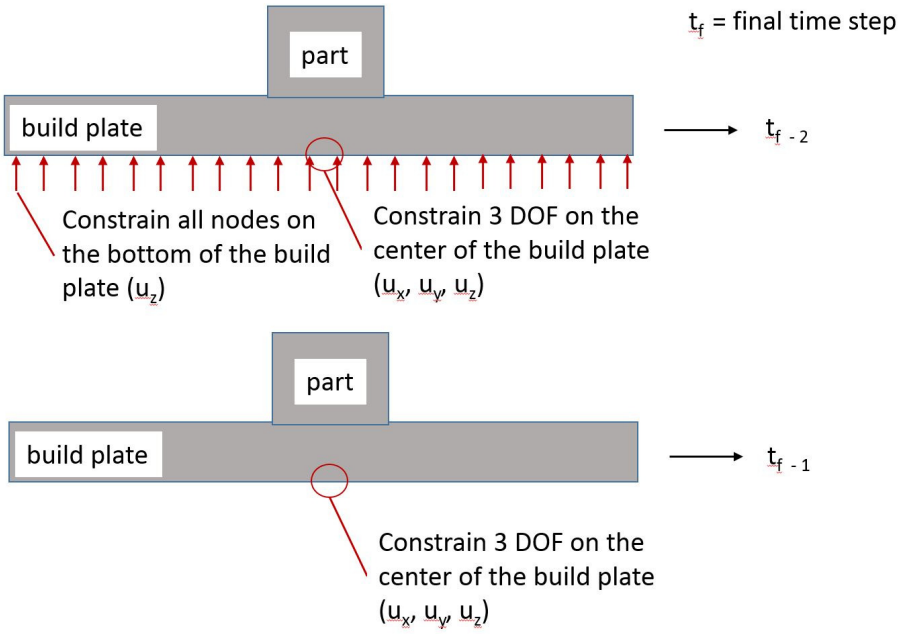


Figure 3.13: BCs assigned by *FSUB

order of removal, so that first supports are removed while the part is still on the build plate, and then the component is removed from the build plate.
Note: When using this functionality, warped STLs for the supports cannot be generated.

*FSBT: Floating SuBstrate Type

*FSBT
i1

*FBST controls the boundary conditions during the mechanical analysis before and after the simulated removal of the built component and the build plate. i1 is set to either 0 or 1, 0 being the default.
When i1 is set to 0:
Before build plate removal: The bottom of the substrate is fixed in z and a small circle in the center of the substrate base is fixed in x,y, and z.
After build plate removal: A small circle in the center of the build plate base is fixe in x,y, and z.

When i1 is set to 1:
Before build plate removal: The bottom of the substrate is fixed in z and a single element in the center of the substrate base is simply supported.
After build plate removal: A single element in the center of the substrate base is simply supported.

Required cards: *FSUB

*STLF: STL File

*STLF
a1
a2
a3
...
a1: a*80: STL file name

This card is used for Powder-Bed Part-Level analyses and allows the user to select a .stl file to use. The mesh will be automatically generated from the file.
Multiple STL files may be imported using this card. Assign material properties and structure type using *STLM. Set the gap tolerance using *STOL. If there are overlaps between the STL files then they are meshed by order of listing.

*NTFE

*NTFE

From version 2018.2 onward when source STL files are determined to have non-manifold edges the solver applies an automatic repair scheme. This uses the default repair module from Netfabb. Using *NTFE switches the repair script to the extended Netfabb repair option. Use this in cases where the default repair module has failed to create a meshable geometry.

*NTFT: Non-maniFold repair Tolerance

*NTFT
r1

From version 2018.2 onward when source STL files are determined to have non-manifold edges the solver applies an automatic repair scheme. Adjust the repair tolerance, r1, to adjust the repair script operation. By default r1 is 0.01 mm^3. This works with both the default repair script, which is automatic, or the manually chosen extended repair, enabled using *NTFE.

*STLM: multiple STL Mapping

*STLM
i1, i2, i3, r1
i5, i6, i7, r2
...
i1, i5: i*4: Configuration id: 1=part, 2=build plate, 3= support structure
i2, i6: i*4: PRM number
i3, i7: i*4: Material ID number
r1, r2: r*8: Volume fraction

This card assigns structure type, prm file, material id, and volume fraction. The configuration id number specifies what type of body the imported STL is, build part or support structure. The PRM number allows multiple PRM files to be used, which allows for modeling using different parameters

on different parts. This number should correspond to the location in the list provided in *PBPF. The material id set the material properties for each STL component using the values in *MATI. Using a build plate STL allows for simulations to capture the exact geometry of the substrate used during powder bed fusion.
Support structure volume fraction is calculated by dividing the original support structure volume by the homogenized (also known as block type, solid volume, or shrink-wrapped) support structure. The volume fraction is used to scale properties for support structure STL files, using a value of 0 to 1. Homogenized lattice geometries can be modeled similarly using a volume fraction equal to the lattice volume divided by the original non-lattice part volume. The volume fraction should be set to 1 for solid part files. When using homogenized support or lattice STL files, adjust the volume fraction so that it matches the ratio of lattice structure volume built to the total volume the support structure encompasses.
Required cards: *STLF

*STLC: STL Contact volume fractions

*STLC
r1
STL file in STLM 1 r2
STL file in STLM 2 ...

*STLC is used to assign a different volume fraction for the interfaces between parts. This is used to improve the modeling of support structure failure, by allowing users to specify the volume fraction of the support structure teeth, which is where support structure failure is modeled. The volume fractions will follow the order set in *STLM. Component STL files must be given an interface fraction even though the values will not be used.
Required cards: *STLM, *STLF

*STIC: STl Individual Contact volume fractions

*STIC
i1, r1: STL index 1, volume fraction 1
i2, r2: STL index 2, volume fraction 2
...
*STIC is used for the same purpose of *STLC, but is used to specify volume fractions just for the STLs chosen, not requiring every STL to be given a contact volume fraction.
Required cards: *STLM, *STLF

*STLH: STL Homogenization - Tech Preview

*STLH
i1, r1,i2,r2

i1: i*4: STL reference, equivalent to line number in *STLF and *STLM
r1: r*8: Alpha radius for homogenization, in mm
i2: i*4: Volume fraction control option
r2: r*2: Structure thickness in mm, for i2=2, must have dummy value for other i2 options

The *STLH card allows users to automatically homogenize any STL file. Homogenization will create a solid volume equivalent geometry and then scale the material properties based upon the volume fraction. The volume fraction is the ratio of the volume of the original part over the volume of the total homogenized volume. This process can speed up simulations significantly by allowing coarser meshing settings to be chosen. This can be applied to both sacrificial support structures and lattice type components. Any cavity of the STL smaller than the Alpha radius will be homogenized. There are three options to specify the volume fraction:

- i2 = 0 : Use volume fraction set in *STLM
- i2 = 1 : Calculate volume based upon STL volume. Use only for closed, volumetric, and manifold STL files.
- i2 = 2 : Calculate volume using the optional r2 value as the structure thickness
- i2 = 3 : Calculate volume using the laser beam diameter from the PRM file

Required cards: *STLM, *STLF

*STLL: STL homogenization Line splitting

*STLL
r1: r*8: Maximum homogenized STL line length, default value = 0.8*(2*alpha radius)

Using STL homogenization via *STLH users may be confronted with a situation where the alpha radius required to incorporate both ends of an STL line segment are greater than the desired alpha radius, so that either gaps that are desired to be remained are filled in or some source features are homogenized and included in the mesh. *STLL is used to subdivide line lengths exceeding a set threshold. This feature is enabled automatically with default value of r1 set to be 80% of the maximum gap specified by the alpha radius. Use the *STLL card to overwrite the default value.
Required cards: *STLM, *STLF, *STLH

*STL3: cancel STL homogenization Line splitting

*STL3
Use *STL3 to turn off the automatic line splitting described in *STLL.
Required cards: *STLM, *STLF, *STLH

*SPSH: SPlit stl homogenization - Tech Preview

*SPSH
i1, i2

i1: Homogenized STL shell split toggle, 0 or other positive integer
i2: Log verbosity control, 0,1, or 2

Use this card to split multi-entity homogenized STLs into shells during meshing so that each isolated support or lattice structure is segmented into a single entity before homogenization. Shells with less than 4 vertices are ignored. Triangles are considered part of the same shell only if they share edges. This improves meshing of homogenized parts as regions between shells will not be artificially filled in during meshing.

Use any i1 value other than 0 to turn this feature on. A value for i1=0 turns the feature off. Users can control the verbosity of the shell splitting feature by adjusting i2. 0 logs a minimal amount of summary information, like the total number of tets. 1 logs the wall time and number of tets for each of possibly hundreds of shells. 2 will also write a file "results_STLName_shell_XXXX.stl" for each individual shell.
Best Practices: This feature is not compatible with support structures created with raster or contour fragments.
Required cards: *STLM, *STLF, *STLH

*MNHV: Maximum Number of Homogenized Vertices

*MNHV
i1: number of vertices

Homogenization of STLs with large number of vertices seeding problems may occur. For large STLs, you can use the *MNHV card to subsample a given maximum number of vertices for homogenization. By default, a consistent pseudo-random subsample of i1 vertices is used. To turn off pseudo-randomization of STLs, you can use the *NSTP card to use the first i1 vertices.

*NSTP: No STl Pseudo-randomization

*NSTP

Use *NSTP to use just the first il vertices specified by the *MNHV card.

*STOL: multiple Stl TOLerance

*STOL
r1

r1: r*8: Multiple STL gap tolerance
This card is used to prevent holes between parts when using multiple STL files. It should be assigned in the native STL units (e.g. mm, in, m, etc.). This value will expand each STL by the value chosen, r1, so it is best practice to make this as small as possible, without producing any holes. If overlap occurs, the mesh is assigned to the value in *STLM via the order of listing in *STLF.

*STL12: SToL version 1 tolerance

*STL1
This card reverts to the old (Version 2018.1 and previous) STL expander for *STOL.
Required cards: *STOL

*STL2: SToL version 2 tolerance

*STL2
r1, r2
STL tolerances

*STL2 is used in conjunction with the new *STOL algorithm for Versions 2018.2+. The new STL expander preserves angles between faces and edges, which the old algorithm, still accessible using *STL1, did not. The tolerances set by r1 and r2 are to handle near-singular cases. The r1 tolerance is set for a 3D near-singular cases, for nearly parallel planes that meet at a single vertex. The r2 tolerance is set for a 2D near-singular case where two nearly parallel planes meet a a single vertex. The default values for both r1 and r2 are 0.1. Poor tolerances for ill-conditioned STLs can expand the STL in unexpected directions.
Required cards: *STOL

*UNIO: UNIOn of self intersections

*UNIO
By default, self intersections will not be meshed. *UNIO is used to enable meshing of STL self intersections. This does not apply to the intersections of separate STL files.

*ETOL: minimum stl Edge TOLerance

*ETOL

This card uses the minimum triangle edge length as a basis for STL vertex equivalencing.

*EMUL: Etol MULtiplier

*EMUL
r1

This card uses r1 for the *ETOL multiplier, default 0.1.

*BTOL: minimum stl Bounding box TOLerance

*BTOL

This card uses the bounding box as a basis for STL vertex equivalencing.

*BMUL: Btol MULtiplier

*BMUL
r1

The card uses r1 for the *BTOL multiplier, default 1.d-6.

*STLS: STL file Scaling

*STLS
r1
r1: r*8: STL file scaling factor

This option is used to make the .stl file units consistent with other analysis inputs. For example if the STL file is in inches, and the model properties are in mm, r1 should be 25.4

*RSTL: Ray casting STL algorithm switch

*RSTL

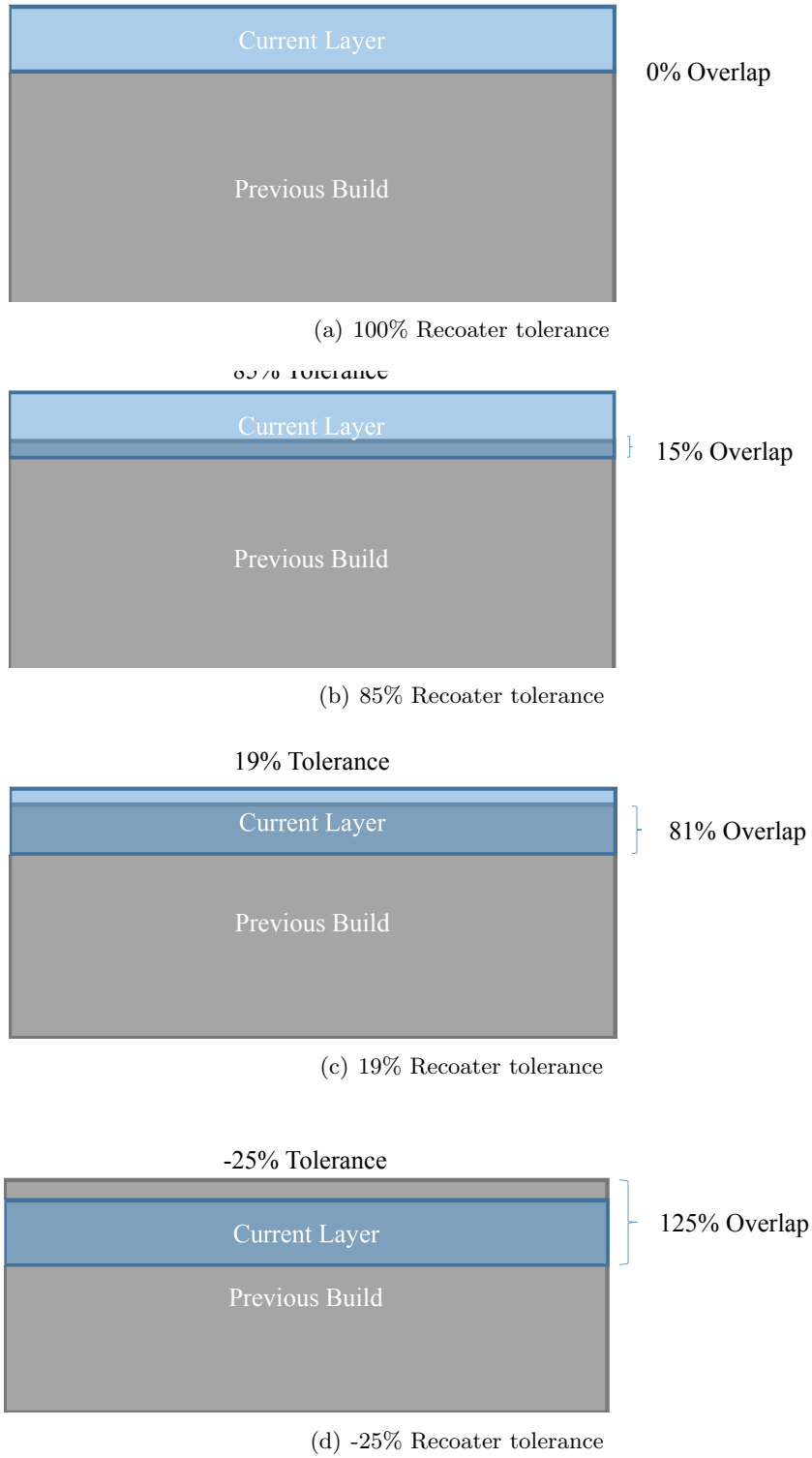
This option reverts to the old ray-casting STL meshing tool.

*RCTR: ReCoaTeR tolerance

*RCTR
r1 [i1]
r1: r*8: Recoater tolerance in %. Default 20%.
i1: i*4: Recoater interference stoppage switch

If the thickness of the powder layer is reduced due to distortion below r1% of the nominal layer thickness, a warning message is generated signalling a possible interference between the recoater blade and the build. If i1, the Recoater interference stoppage switch, is set to any value except for 0, the simulation will abort once the first recoater interference warning is encountered.
Figure 3.14 helps illustrate how recoater interference is calculated. In Figure 3.14(a), there is 0.40 mm allocated for the fusion of the next layer, and there is 0.40 mm on top of the build, which results in a 100% clearance. In Figure 3.14(b) there is 0.06 mm taken up by the build of previous layers, which results in an interferences of 85% recoater interface. At 0.321 mm build interface, there will be 19% recoater tolerance and an warning message will be recorded, as shown in Figure 3.14(c). If the build exceeds the entire build height allocated for the current layer, e.g. 0.50 mm for a 0.40 mm recoater height, which returns an warning and a calculated interference of -25%, which is depicted in Figure 3.14(d).
Along with the log files, appended .out, the text file, file_name_recoater.txt is written after the mechanical simulation completes. This file records the time, layer group number, recoater clearance %, peak height of the deformed model at the top of the layer group, and the nominal recoater height at the top of the layer group. A sample file is shown below:

```
time (s), layer group, recoater clearance (%), top z deformed coord (mm), recoater coord (mm)
  9.024533E+03             1        93.331  1.602668E+00  1.640000E+00
  1.805244E+04             2        89.961  3.204015E+00  3.240000E+00
  2.712766E+04             3        76.513  4.809395E+00  4.840000E+00
  3.624973E+04             4        72.380  6.411048E+00  6.440000E+00
  4.540336E+04             5        73.722  8.010511E+00  8.040000E+00
  5.457021E+04             6        78.515  9.608594E+00  9.640000E+00
  6.375136E+04             7        80.608  1.120776E+01  1.124000E+01
  7.295711E+04             8        82.051  1.280718E+01  1.284000E+01
  8.221741E+04             9        74.441  1.441022E+01  1.444000E+01
  9.153668E+04            10        69.227  1.601231E+01  1.604000E+01
  1.010004E+05            11        -1.821  1.764073E+01  1.764000E+01
  1.106051E+05            12       -24.325  1.924973E+01  1.924000E+01
  1.204665E+05            13         0.818  2.083967E+01  2.084000E+01
  1.308714E+05            14         7.291  2.243708E+01  2.244000E+01
  1.414172E+05            15         4.956  2.403802E+01  2.404000E+01
  1.519087E+05            16        -4.642  2.564186E+01  2.564000E+01
  1.624634E+05            17        -1.487  2.724059E+01  2.724000E+01
  1.728352E+05            18        -2.322  2.884093E+01  2.884000E+01
```



(a) 100% Recoater tolerance

(b) 85% Recoater tolerance

(c) 19% Recoater tolerance

(d) -25% Recoater tolerance

Figure 3.14: Recoater tolerance calculation examples

```
1.830612E+05          19          8.865  3.043645E+01  3.044000E+01
1.932244E+05          20          6.861  3.203726E+01  3.204000E+01
2.032568E+05          21          4.110  3.363836E+01  3.364000E+01
2.133567E+05          22          2.824  3.523887E+01  3.524000E+01
2.229929E+05          23         -9.362  3.684374E+01  3.684000E+01
```

This example file shows that recoater interference is almost certainly going to be a problem for this build, starting from layer 11 onward. This is another feature to help guide the end user towards adding or strengthening support structures, change building parameters, orientation, or part geometry to ensure a useful part can be printed.

*ORCS: Opposite of Recoater Clearance

*ORCS
Using this card outputs a new result to the recoater.txt file which records the ratio of the lowest point of the currently modeled layer group to the layer thickness, reported as a percentage.

*PBCH: Powder Bed CHech matrix size

*PBCH

This card turns on check run or the sparse matrix pre-processing (auxspar) for the entire build mesh at the beginning of the analysis. This feature is useful in calculating the biggest stiffness matrix size before the entire analysis is run.

*PBDL: Powder Bed DwelL time

*PBDL
r1
r1: r*8: number of parts on build plate.
This option automatically calculates, based on the layer volume, the amount of time that it takes to deposit layers in powder-bed processing. The time is added into part scale analyses to allow for the calculation of actual cooling periods. The time is calculated using the following equation:

$$t = \frac{V}{vhd} r1 \tag{3.1}$$

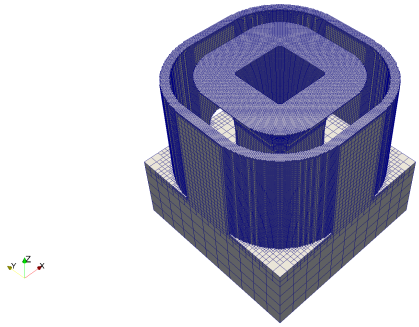
where, t is the added processing time, V is the volume of the added material calculated from the part geometry, v the laser speed, h is the hatch spacing, and d is a single powder layer thickness. Adjusting r1 accounts for the case where a user is simulating fewer parts than what will appear on the actual build plate. For example, if 4 similar parts will be on the actual build plate and the user wishes to simulate the build using an .stl file that contains only 2 of the parts, r1 should be set to 2. This will double the analysis dwell times.
Multi laser system considerations

For multi-laser systems, adjust *PBDL with the number of lasers in use. For example for a large part being built in a 2 laser system, set *PBDL = 0.5, to account for the decreased time. Or when building 12 parts on a 4 laser system, *PBDL = 3.

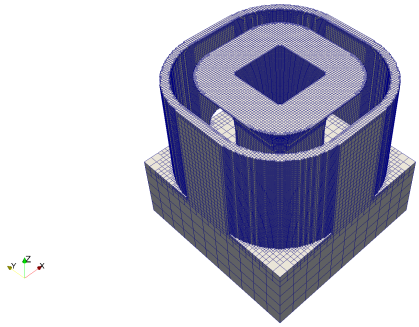
*PBLR: Powder Bed Layer Refinement

*PBLR
i1
i1: i*4: number of generations coarsened per activation layer. Default 0.
This option allows using small element size (defined by *PBPA) to account for fine geometry details or thin sections, and activating layers thicker than the finest element size. The activation thickness is equal to $n \cdot t \cdot 2^{i1}$, where n is the number of layers per group and t is the powder layer thickness (i2 and r1 values in *PBPA card). The effect of the *PBLR card is illustrated in Figure 3.15, where in Figure 3.15(a) it is shown that the added group of elements is composed of fine elements without the *PBLR card. The adaptive coarsening allowed by *PBLR is shown in Figure 3.15(b) and Figure 3.15(c). Note that single core run times can be quartered between the No *PBLR and *PBLR = 2 simulations, however this coarsening will reduce the accuracy of the simulation.

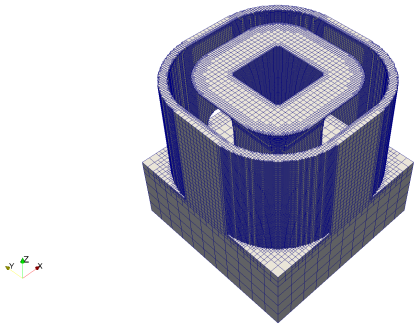
Best Practices: *PBLR = 1 should achieve a fair compromise between speed and accuracy for most geometries. Larger geometries should use values of *PBLR = 2, extremely large builds,*PBLR = 3. When performing quantitative stress via *PPLA or heat treatment simulations using *STRF, lower values of 0 or 1 should be used to ensure the mesh is adequately converged. Good FE practice requires a mesh convergence study for each model.



(a) No *PBLR, *PBPA = 5, Run time = 13965 s



(b) *PBLR = 1,*PBPA = 5, Run time = 6462 s



(c) *PBLR = 2,*PBPA = 5, Run time = 3213 s

Figure 3.15: Illustration of *PBLR and its effect upon single core run time.

*GTAB: Generate tables

*GTAB

This option allows the user to output a process parameter (.prm) file from a moving source analysis. This .prm file is required to run Part-Level Powder-Bed analyses.
This option must be used in both the thermal and mechanical analysis when generating .prm files.
To get usable prm files, you must run the simulations using the prm_gen command

*NAPL: Use New method for *GTAB analyses

*NAPL

This card directs the code to use a new method for *GTAB analyses. The old method was stretching free faces of the modes to zero out-of-plane displacement at the last time increment. When *NAPL is used, the faces remain free.
Required Cards: *GTAB

*HPRM: Hidden process PaRaMeters

*HPRM

This card turns off the default option which outputs the processing parameters in log files, which can be used to protect proprietary processing parameters.

*HMAT: Hidden MATerial properties

*HMAT

Material properties are written by default into the PRM file. These are then read by the solver during Part Scale analysis, so material properties no longer have to be specified again in the part scale input files. The material properties used are written to the output file during part scale simulations. Using the *HMAT card will prevent properties from being written to the output file.

3.7 Material Property Definition

*MATE: Material Property Block

*MATE
*MATI
i1
CARD
...
*MATE
*MATI
i2
CARD
...

i1: i*4: Material ID
i2: i*4: Material ID
*CARD: property card blocks such as *COND, *ELAS, ...
...
Material properties are defined within blocks starting with the *MATE card and ending when another card other than material property is used. Properties per material ID are defined within each block starting with *MATI followed by an an integer (i1,i2, ...) corresponding the Netfabb Simulation auto generated mesh or the material ID used in the PATRAN neutral input file. Internal checks ensure that the necessary properties have been defined by the *MATE card and that they fall within a valid range.
With the inclusion of material properties in the PRM files, it is not necessary to prescribe material properties for the deposition material during part scale simulations. If material properties are included in a part scale input file, they will be superseded by the properties in the input file, as specified by *STLM and *PBPF. Use *MATE in conjunction with *PBS2 to prescribe different properties from the deposition properties, to the build plate.

*COND: Thermal Conductivity

*COND
r11, r12
r21, r22
...
r11: r*8: Thermal conductivity value at temperature r12
r21: r*8: Thermal conductivity value at temperature r22
...
This card specifies temperature dependent thermal conductivity.

*DASH: DASHpot Stiffness

*DASH
r11, r12
r21, r22
...
r11: r*8: Spring constant at displacement r12
r21: r*8: Spring constant at displacement r22
...
This card defines displacement dependent stiffness for point elements. This option is useful to model workpieces constrained by non-ridged supports. *DASH is used within the *MATE block.

*DSHP: DaSHpot Pretension

*DSHP
r1
r1: r*8: Pretension of spring

This card allows the definition of pre-tension for a spring defined by the *DASH card. The resulting spring force equals $f = k(u)(u - p)$. Where, f is the spring force, k is the spring stiffness (defined by *DASH), u the displacement, and p is the spring pre-tension defined by the r1 value in the

*DSHP. *DSHP is used within the *MATE block.
Required Cards: *DASH

*DENS: Density

*DENS
r1

r1: r*8: Density value

This card specifies temperature independent density in kg/mm^3.

*ELAS: ELAstic Modulus

*ELAS
r11, r12, r13
r21, r22, r23
...
r11: r*8: Elastic Modulus value at temperature r13
r12: r*8: Poisson's ratio value at temperature r13
r21: r*8: Elastic Modulus value at temperature r23
r22: r*8: Poisson's ratio value at temperature r23
...
This card specifies temperature dependent Elastic Modulus in MPa for temperatures in °C.

*EMIS: EMISsivity

*EMIS
r11, r12
r21, r22
...
r11: r*8: Emissivity value at temperature r12
r21: r*8: Emissivity value at temperature r22
...
This card assigns temperature dependent emissivity to account for heat loss via thermal radiation. For auto generated meshes (using *AUTM), radiation will be applied to all free surfaces. For manually generated meshes (using *INPU), if no surface convection is defined in the Patran input file, no radiation is applied even if the *EMIS card is used.
A convection with zero convection coefficient can be applied on surfaces where there is no surface convection (e.g. in a vacuum).

*EXPA: thermal EXPansion coefficient

*EXPA
r1
r11, r12
r21, r22
...

r1: r*8: Reference temperature
r11: r*8: Thermal expansion value at temperature r12
r21: r*8: Thermal expansion value at temperature r22
...
This card specifies the temperature dependent thermal linear coefficient of thermal expansion.

*LATE: LATEnt Heat

*LATE
r1, r2, r3

r1: r*8: Latent heat (energy/volume)
r2: r*8: Solidus temperature
r3: r*8: Liquidus temperature

This option allows for the inclusion of the latent heat of fusion in a thermal analysis. Latent heat is specified as J/kg at Liquidus and Solidus temperatures in °C.
Best Practices:
Widening the Solidus to Liquidus temperature range can improve convergence and reduce run times while having minimal effect on the analysis results. It is best to reduce the lower temperature by 100 °C and increase the higher temperature by °C to avoid such convergence issues caused by the extreme non-linearity introduced by the inclusion of Latent heat in thermal analyses.

*PLAS: PLASticity properties

*PLAS
i1
r11, r12, r13, r14, ..., r1n,
r21, r22, r23, r24, ..., r2n,
...
n=2*i1+1
i1: i*4: number of (yield strength, equivalent plastic strain) points
r11: r*8: Yield strength for equivalent plastic strain r12 at temperature r1n
...
r13: r*8: Yield strength for equivalent plastic strain r14 at temperature r1n
...

- Isotropic plastic hardening is assumed.
- r12, r22, r32,.. should be zero.
- Perfect plasticity is assumed beyond the last (yield strength, equivalent plastic strain) point.
- If i1=1, the response is elastic-perfectly plastic (no hardening).

Yield strength is input in terms of MPa.
Best Practices: Unless *PPLA is used, do not include *PLAS in Part-Scale Powder-Bed Simulations as this behavior is already accounted for in the PRM generation phase and will incur lack of convergence. When performing part-level plasticity analyses using *PPLA ensure there are

*PLAS values assigned to the build plate properties as well, if a separate build plate *MATE and *MATI are used.
Figure 3.16 shows an example of fitting a stress-strain curve using the *PLAS card using 1-4 values. Using just 1 value assumes perfectly plastic behavior. Using 2 points along the stress strain curve will create a bilinear map of plastic properties, one line between the yield point and the second point, and then a region of perfect plasticity extending from the second point. This is the most common method to model the plastic behavior, and has been shown to be effective for various metals. Additional points may be used to further describe the stress-strain curve as desired by the end user. The more detailed the stress-strain behavior is defined in the mechanical model, the slower the convergence will be each step.

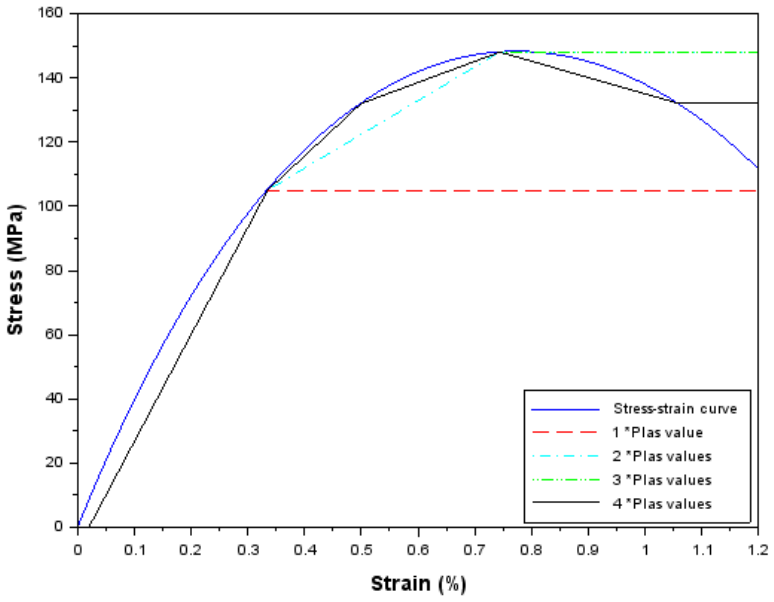


Figure 3.16: Approximating the stress-strain curve using 1-4 values in the *PLAS card

*SPEC: SPECific Heat

*SPEC
r11, r12
r21, r22
...
r11: r*8: Specific heat value at temperature r12
r21: r*8: Specific heat value at temperature r22
...
This option assigns temperature dependent specific heat in units of J/kg/°C.

*MLTT: MeLTing Temperature

*MLTT
r1

r1: r*8: Melting temperature, in °C.

- When the temperature exceeds r1, material melting is simulated.
- If *MLTT is defined outside the *MATE block, the same value is used for all materials.

*SRLX: Stress ReLaXation temperature

*SRLX
r1, [r2]

r1: r*8: Stress relaxation temperature, in °C

This optional material property is used to relax stresses during processing due to material dependent phenomena.

- When the temperature exceeds r1, all elastic, plastic, and thermal stress and strain components are set to zero.
- If *SRLX is defined outside the *MATE block, the same value is used for all materials.

Best Practices:
This option should be used for mechanical analyses of Ti-6Al-4V with r1 set to the stress relaxation temperature of 690 °C.

*ANNL: ANNeaL temperature

*ANNL
r1

r1: r*8: Anneal temperature

- When the temperature exceeds r1, plastic strain is set to zero.
- If *ANNL is defined outside the *MATE block, the same value is used for all materials.

This option incorporates material annealing (erasing of dislocations at high temperatures) at temperature r1 (in °C) into the mechanical analysis.

*EVAP: EVAPoration temperature

*EVAP
r1

This option specifies the evaporation temperature (in °C) for the deposited material and is listed outside the material block.

*TCUT: Temperature CUT off value

*TCUT
r1

r1: r*8: temperature cut off value

- Used in mechanical analyses to neglect temperatures higher than melting.
- Temperatures above r1 (°C), are clipped to the r1 value.
- This card can minimize excessive plasticity and improve convergence for mechanical analyses.
- **Currently *TCUT is defined outside the *MATE block and the same value is used for all materials in the analysis.**

*NMAT: Number of MAterials

*NMAT
i1

*NMAT is used to set the maximum number of materials the solver can process. The default value is 100. ***NMTT must be the first card in the input file to work.** If both *NMAT and *NMTT are used their order does not matter but they must be the 1st two cards in the input file.

*NMTT: Number of MaTerial Temperatures

*NMTT
i1

*NMTT is used to set the maximum number of temperature values the solver can process for the material property cards, like *COND and *ELAS, as well as boundary conditions, like *CONV and *FIXR. The default value is 100. ***NMTT must be the first card in the input file to work.** If both *NMAT and *NMTT are used their order does not matter but they must be the 1st two cards in the input file.

*MNAM: Material NAMe

*MNAM
i1
a1, Optional, only used if i1=0

i1: Material type: 0 for named (library) material; 1 (or other value) for custom material.
a1: Material name. Only present if i1 = 0.

*MNAM, used in the thermal PRM input file, writes the material name the resulting PRM file. This allows Simulation Utility for Netfabb to read and display this value in the Processing Parameters library, which allows users to create new PRM files rapidly from existing PRMs.

*PTYP: Powder material TYPe

*PTYP
i1

i1: powder material type. 0 for automatically calculated; 1 for custom scaling factors; 2 (or other value) for fully custom.

*PTYP, used in the thermal PRM input file, encodes the Powder Material type to the resulting PRM file. This allows Simulation Utility for Netfabb to read and display this value in the Processing Parameters library, which allows users to create new PRM files rapidly from existing PRMs.

3.8 Boundary Conditions

*AUBC: AUtomatic Boundary Conditions switch

*AUBC

This option can be used in conjunction with *ADAP for manually generated meshes using *INPU. Boundary conditions are automatically applied to all automatically generated (using *AUTM) meshes.
When *AUBC is used, the surface convection defined by the *CONV and the energy defined by the *LSRF card is used without boundary condition definition in PATRAN.
Required Cards: *INPU

*RAMP: Ramping of boundary conditions

*RAMP
ID
i1, i2
r11, r12
r21, r22
... *ID
i1, i2
r11, r12
r21, r22
...
i1: r*4: boundary condition ID
i2: r*4: load case number

r11: r*8: multiplier at time r12
r21: r*8: multiplier at time r22
...
Boundary condition ID's:
6 element pressure
7 nodal force
8 nodal displacement
10 nodal temperature

15 nodal heat source
16 distributed heat source
17 convection
18 (radiation emissivity) not implemented
99 point (dash) elements)

For point elements (i1=99), i2 is the material ID of the element.
This card allows for boundary conditions to be scaled with respect to time.

3.8.1 Temperature Boundary Conditions

*AMBI: Ambient Temperature

*AMBI
r1

This option sets the ambient temperature for thermal and mechanical analyses in °C.

*INIT: INItial Temperature

*INIT
r1

r1: r*8: initial temperature value

This card defines the initial temperature for both thermal and mechanical analyses, in °C. If *INIT is not defined, the ambient temperature defined by *AMBI is used as the initial temperature. If *INIT is used, then by default this is the final temperature the part is returned too after the simulation has completed.
Best Practices:
When using *INIT, it is suggested to use *FINT to force the part to cool down to room temperature, as this is most representative of reality.

*FINT: FINal Temperature

*FINT
r1

r1: r*8: final temperature value

This temperature, in °C, is used in the extra cooling *COOL increment in mechanical analyses. If *FINT is not defined, the initial temperature defined by *INIT is used.

*TAMB: Time dependent AMBient temperature

*TAMB
r11, r12
r21, r22
...

r11: r*8: Ambient temperature value at time r12
r21: r*8: Ambient temperature value at time r22
...
When *TAMB is used, any values defined by *AMBI are ignored.
This option allows for the ambient temperate, in °C, during a thermal or mechanical analysis to be dependent on time, to simulate the change of temperature within a machine chamber during processing.

3.8.2 Convection Boundary Conditions

*CONV: CONVection coefficients

*CONV
r11, r12
r21, r22
...
r11: r*8: Convection at temperature r12
r21: r*8: Convection at temperature r22
...
This allows for the definition of a temperature dependent convection in W/mm^2 °C for temperatures specified in °C. Convection is automatically applied to the free surfaces of *AUTM auto generated meshes. These values are also used at the interface of quiet and active elements (unless *DDMP is turned on).

*CCNV: Calculated CoNVection coefficients

*CCNV

When enabled, this card will automatically apply predetermined convection coefficients to the external surfaces of the modeled component based upon the thermal conductivity of the material. The function used has been constructed comparing convection and powder boundary conditions, the values of which are plotted in Figure 3.17. Using this card should yield a thermal history close to that calculated when using powder elements with *+PDR, but without the computational overhead of meshing the powder and modeling thermal losses into it directly. The convection which is value calculated and applied to the model is recorded in the thermal log file.

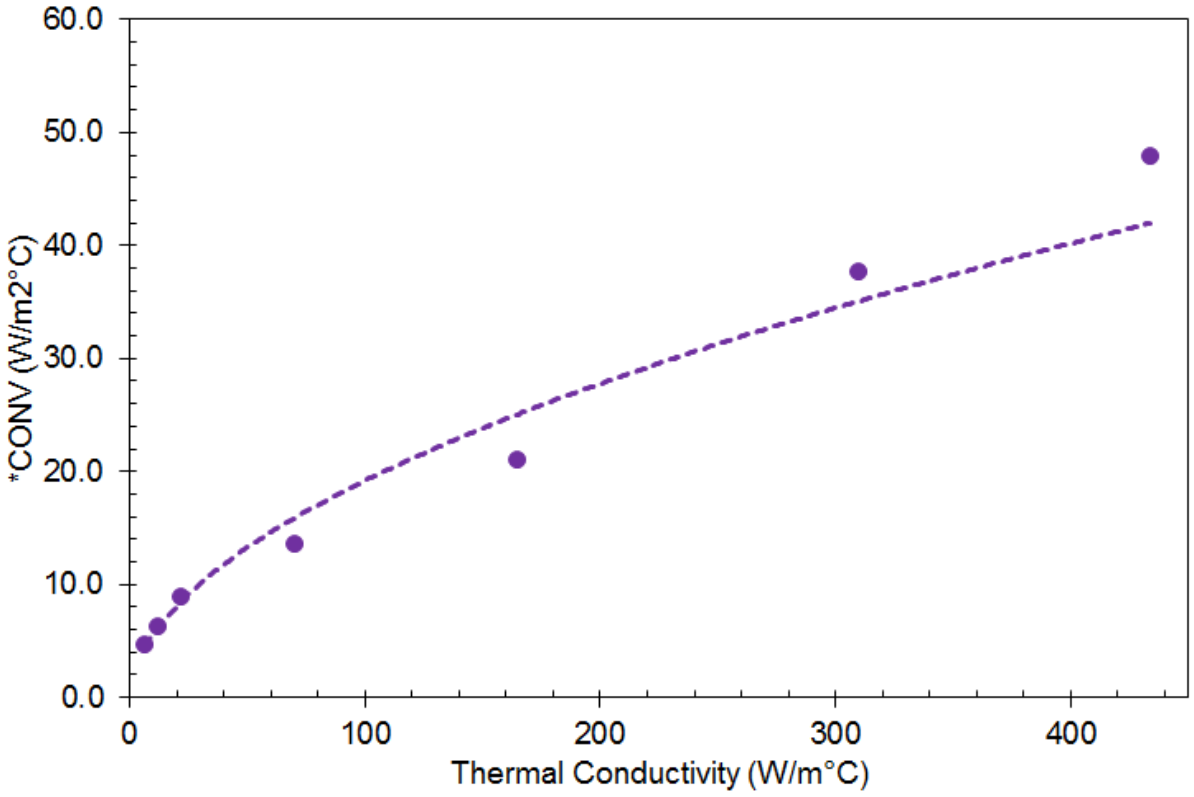


Figure 3.17: Automatically calculated heat loss coefficients based upon thermal conductivity

*TCNV: Thickness dependent CoNVection coefficients

*TCNV
r11, r12
r21, r22
...
r11: r*8: Convection at thickness r12
r21: r*8: Convection at thickness r22
...
*TCNV allows the user to specify thickness dependent convection values. This will be applied to parts, homogenized parts, and supports. Convection values for the build plate will be those specified by the *CONV card.

*PBCT: Powder Bed Convection Top

*PBCT
r1, r2

r1: r*8: Top surface convection coefficient r2: r*8: Sink temperature for top surface convection
*PBCT specifies the convection value and sink temperature for the top layer during each part-level powder bed simulation increment. This applies to the top of the part, and if enabled via *+PDR,

the top layer of loose powder elements. *PBCT is static, not temperature dependent. *PBCT overrides any *CONV value set. Consult Figure 3.18(a) and Figure 3.18(b) to see how this card is applied for the powder and no powder cases.

*PBCP: Powder Bed Convection Powder sides

*PBCP
r1, r2

r1: r*8: Powder sides surface convection coefficient r2: r*8: Sink temperature for powder sides convection
*PBCP specifies the convection value and sink temperature for the sides of the powder during each part-level powder bed simulation increment. This card requires loose powder elements to be turned on by *+PDR to be used. *PBCP is static, not temperature dependent. *PBCP overides any *CONV value set. Consult Figure 3.18(a) and Figure 3.18(b) to see how this card is applied for the powder and no powder cases.
Required Cards:*+PDR

*PCSS: Powder Bed Substrate Sides convection

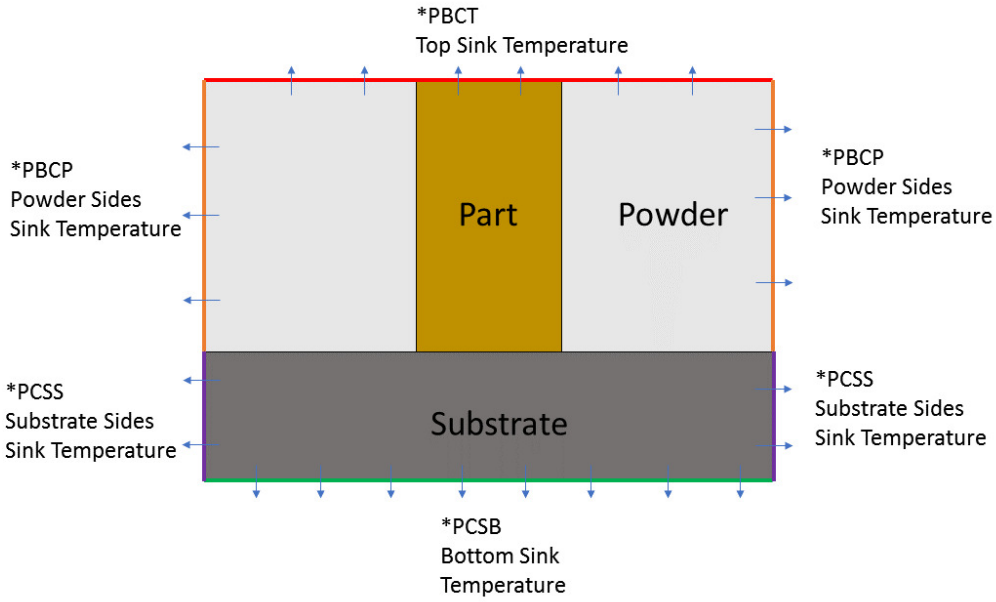
*PCSS
r1, r2

r1: r*8: Substrate sides convection coefficient r2: r*8: Sink temperature for substrate sides convection
*PCSS specifies the convection value and sink temperature for the sides of the substrate or build plate during each part-level powder bed simulation increment. *PCSS is static, not temperature dependent. *PCSS overides any *CONV value set. Consult Figure 3.18(a) and Figure 3.18(b) to see how this card is applied for the powder and no powder cases.

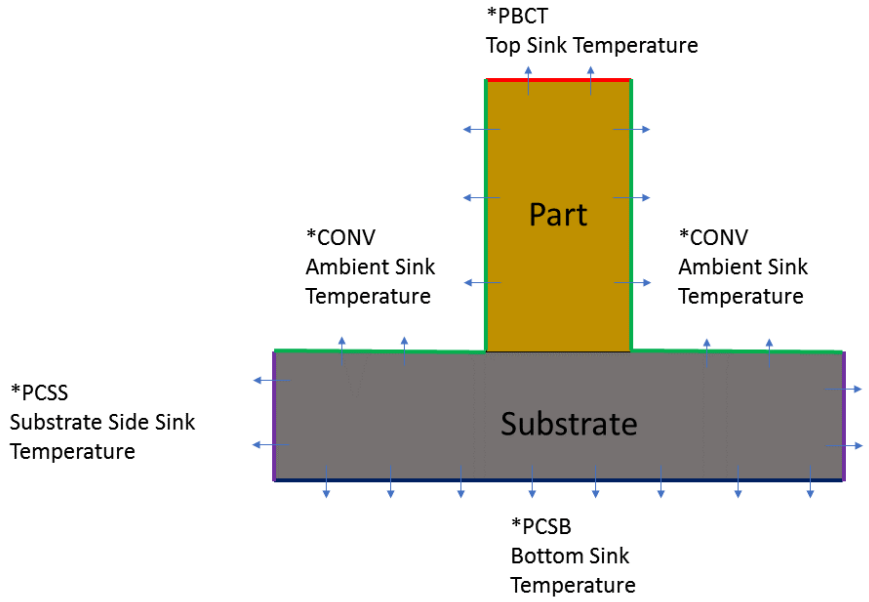
*PCSB: Powder Bed Substrate Bottom convection

*PCSB
r1, r2

r1: r*8: Substrate bottom convection coefficient r2: r*8: Sink temperature for substrate bottom convection
*PBSB specifies the convection value and sink temperature for the bottom of the substrate or build plate during each part-level powder bed simulation increment. *PBSB is static, not temperature dependent. *PBSB overides any *CONV value set. Consult Figure 3.18(a) and Figure 3.18(b) to see how this card is applied for the powder and no powder cases.



(a) Local convection values with loose powder enabled



(b) Local convection values without loose powder

Figure 3.18: The effect of using local convection cards with or without loose powder enabled

*LENS: LENS® convection values

*LENS
r1, r2, [r3]
r1: r*8: sphere radius. Default 0.d0
r2: r*8: local convection. Default 0.d0
r3: r*8: time gas is turned off. Default 1.d50

This card applies an additional component to the applied convection specified by *CONV in thermal analyses. This is used to simulate the the forced convection due to the forced gas flows in the LENS® process or similar Laser based Directed Energy Deposition machines [2].
*LENS models the region of forced convection as a sphere whose center is co-located with the center of the simulated heat source as controlled by *LSRF, of a radius r1 in mm, of a value r2 in W/mm^2, which is cut off at time r3 in seconds.
A custom distribution of the forced convection can be programmed through the lensconvS subroutine in the user.f file.

3.8.3 Heat Source Boundary Conditions

*GOLD: Values for GOLDak's heat source model [3]

*GOLD
r1, r2, r3, r4, r5, r6

The heat source definition is illustrated in Figure 3.19.
r1: r*8: efficiency. Default 1.d0
r2: r*8: b axis multiplier. Default 1.d0
r3: r*8: c1 axis multiplier. Default 1.d0
r4: r*8: c2 axis multiplier. Default 4.d0
r5: r*8: f1 factor. Default .6d0
r6: r*8: f2 factor. Default 1.4d0

- The a axis dimension is set as the melt pool radius (typically about equal to the laser spot size) as defined in *LSRF
- r1 should be set to the absorption efficiency value.
- The b axis dimension is set as r2*a
- The c1 axis dimension is set as r3*a
- The c2 axis dimension is set as r4*a

Best Practices:
For moving-source powder bed, LENS® or other thin layer deposition, the b parameter, which controls the Z axis of the double ellipsoid volume of the heat input model, should be set to r2 = 0.6, which should give a fairly accurate representation of the melt pool, as described in [3] and other modeling literature.
For thicker depositions, r2 should be adjusted so that the laser depth and layer height are identical, e.g.

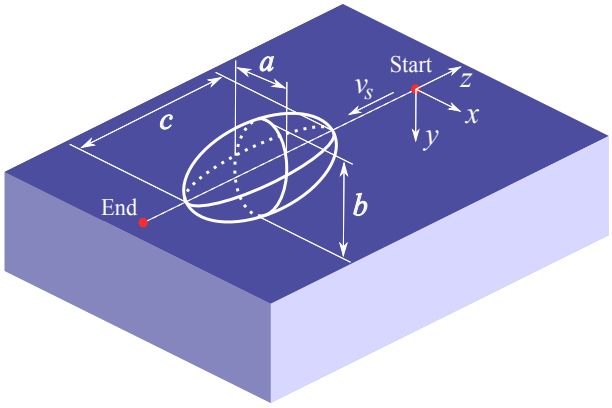


Figure 3.19: Schematic of the Goldak heat source

r2 = layer height/laser radius
This is performed to ensure the entire deposition is melted. If there are unactivated elements or layers in the simulation results, check to see if the r2 value satisfies the above condition.

*3PLP: Improved Goldak accuracy card

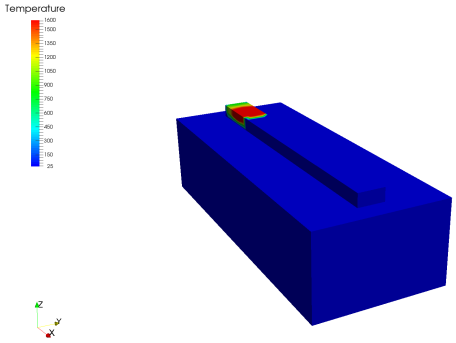
*3PLP

Using this card will improve the accuracy of the Goldak heat input model. Use this card when using the simulation tool to investigate melt pool and related phenomena. Note that using this card will require a simulation to go through more increments and will take more time to complete.

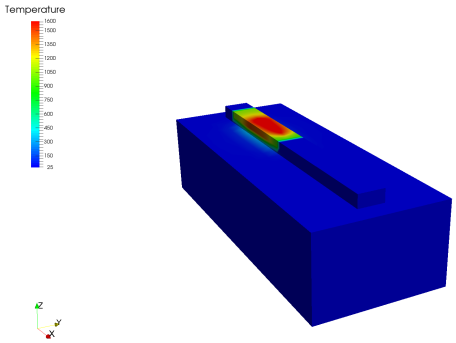
*LINQ: Line heat input (Q) model

*LINQ

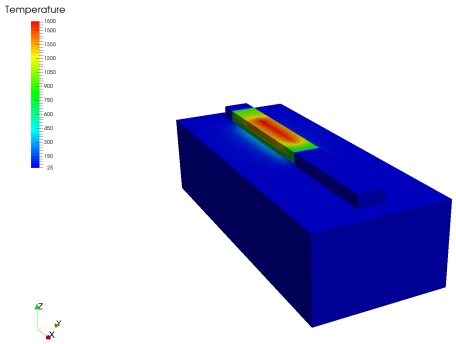
This option activates a line heat input model using an averaged heat source over a line. The length of the line is determined by the *TAUT card. For example, if for a value of 10 is specified by *TAUT, the length of the line is going to be 10 times the radius of the heat source as defined by the *LSRF card. When *LINQ is used, the content of *GOLD is still needed to compute the shape of the heat source distribution. Currently this option is only compatible with thermal analyses.
An illustration of the effect of using *LINQ is given in Figure 3.20 for the same simulation without using *LINQ (Figure 3.20(a)).
Required Cards: *GOLD, *TAUT, *LSRP or *LSRF.



(a) *LINQ = off



(b) *LINQ = 5



(c) *LINQ = 10

Figure 3.20: Line heat input examples using *LINQ

*GTOL: Tolerance values for source heating model

*GTOL
r1, r2

r1: r*8: line extension multiplier. Default 0.d0
r2: r*8: power tolerance. Default 1.d-10

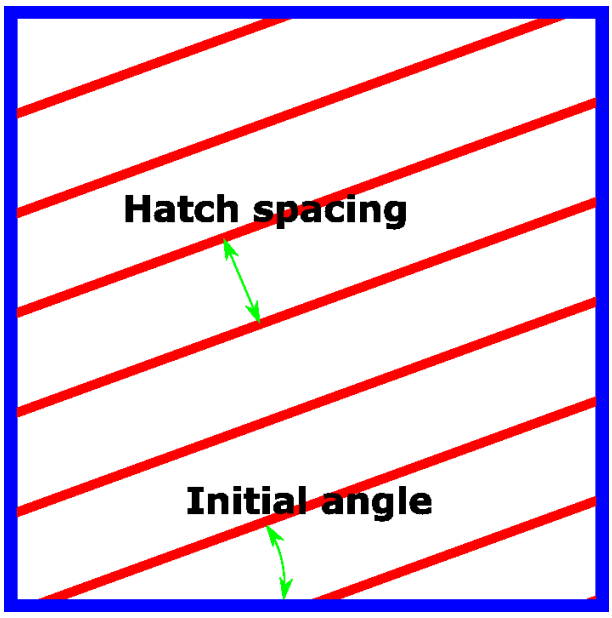
r1 controls a small extension to the heating lines. If r1=1, heating starts c1 before the line start and ends after c2 after the line end (see *GOLD card).
r2 is a tolerance for element activation. If the power in a heating line is less than this value, elements are not activated. This value is useful when lines are added with zero power to model the moving forced cooling using the *LENS card when the laser power is off.

*LSRP: LaSeR Path generation

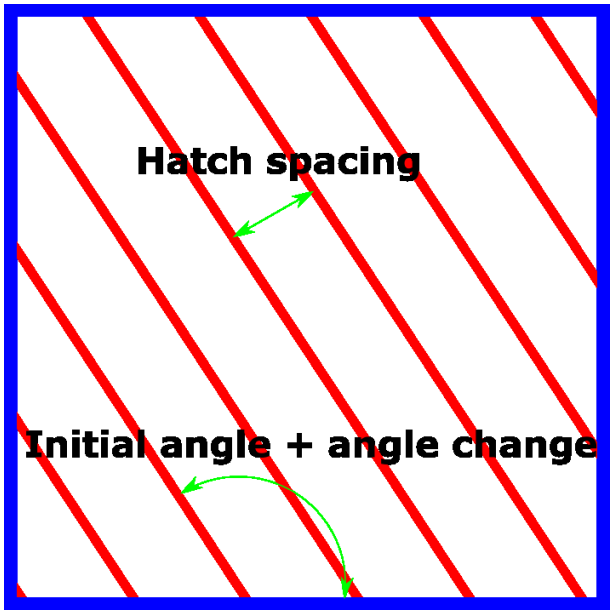
*LSRP
r1, r2, r3, r4, r5, r6, i1, r7, r8, r9, [r10]

r1: r*8: power
r2: r*8 radius of melt pool
r3: r*8: travel speed
r4: r*8: layer thickness
r5: r*8: hatch spacing (gap width)
r6: r*8: dwell time from layer to layer
i1: i*4: number of layers
r7: r*8: initial vector angle [No longer editable]
r8: r*8: vector angle change from layer to layer
r9: r*8: x width of part
r10: r*8: y width of part (if omitted set to r9)

This card generates a laser path for a rectangular patch with an area of r9 x r10. The path is stored as text file in <filename>.lsr, and used for the analysis. The value of the top substrate z coordinate is defined by the DDM!. The laser path extends out of the part by a distance equal to the melt pool radius (r2).
Note the initial vector angle is no longer editable. If any value besides the default value of 11.5 degrees is used, it will be reset to the default value.
Typically the radius of the melt pool (r2) is about twice the radius of the laser beam.
Figure 3.21 depicts an example schematic of the first two build layers using paths generated using *LSRP.



(a) Layer 1



(b) Layer 2

Figure 3.21: Illustration of two build layers automatically generated paths using *LSRP

*LSRF: Moving heat source definition file

*LSRF
a1: a*80: File name with content of laser lines

This option defines individual laser paths by reading a file, generated by the user, with the following format:
r11, r12, r13, r14, r15, r16, r17, r18, r19, r1(10), r1(11), r1(12), r1(13)
r21, r22, r23, r24, r25, r26, r27, r28, r29, r2(10), r2(11), r2(12), r2(13)
...
r11: r*8: power
r12, r13, r14: r*8: vector of heat source direction
r15, r16, r17: r*8: start point
r18, r19, r1(10): r*8: end point
r1(11): r*8: melt pool radius
r1(12): r*8: velocity
r1(13): r*8: start time
...
The heat source path is composed of linear segments. Paths must be in the XY plane only.

*LSR2: LaSeR file format 2

*LSR2
a1: a*80: File name with content of laser lines

This card references the laser vector file specified by a1. Use this input for moving source simulations which have linearly varying power across individual laser passes. The file format for the laser vector file is:
r11, r12, r13, r14, r15, r16, r17, r18, r19, r1(10), r1(11), r1(12), r1(13),r1(14)
r21, r22, r23, r24, r25, r26, r27, r28, r29, r2(10), r2(11), r2(12), r2(13),r2(14)
...
r11: r*8: Start power
r12: r*8: End power
r13, r14, r15: r*8: vector of heat source direction
r16, r17, r18: r*8: start point
r19, r1(10), r1(11): r*8: end point
r1(12): r*8: melt pool radius
r1(13): r*8: velocity
r1(14): r*8: start time

3.8.4 Mechanical Constraints

*NBLT: No BoLTs

*NBLT
*NBLT mimics the mechanical constraints of a powder bed build plate being simply supported on the build elevator, without being bolted down. This card applies simple supports on the base of the build plate at the center. This allows the build plate to deform during laser processing.

*FIXC: FIXture Circular

*FIXC
r11, r12, r13
r21, r22, r23
.... r11: r*8: Radius r12: r*8: x-coordinate r13: r*8: y-coordinate
This card is used in conjunction with *AUTM, or *STLF to define the circular fixtures on the bottom of the buildplate. All nodes within the defined circles are fixed in all 3 axes. Figure 3.22 illustrates the option.

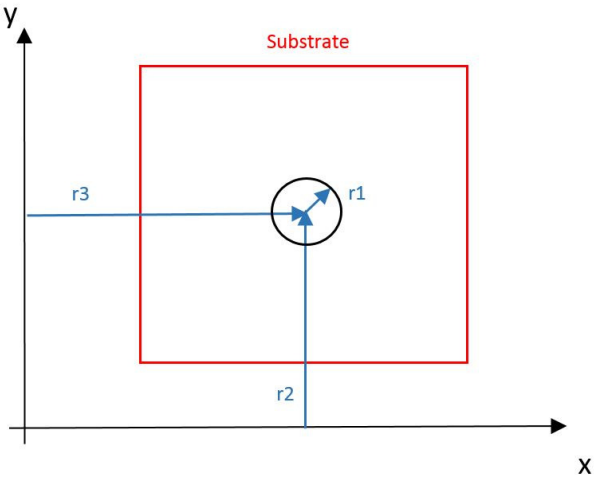


Figure 3.22: Positioning of a circular constraint

Best Practices:
Use *FIxC if practical to avoid overconstraining the part.

*FIxC: FIxture Circular

*FIxC
r11, r12, r13, i11, i12, i13
r21, r22, r23, i21, i22, i23
....
Where:
r11: r*8: Radius r12: r*8: x-coordinate r13: r*8: y-coordinate i11: i*4: x-DOF [0 for free, 1 for fixed] i12: i*4: y-DOF [0 for free, 1 for fixed] i13: i*4: z-DOF [0 for free, 1 for fixed]
This card is similar to *FIXC, but it allows selecting individual degrees of freedom to be fixed. 6 DOF are required to be constrained to prevent rigid body motion during the process simulation.

*FIXR: FIXture Rectangular

*FIXR
r11, r12, r13, r14
r21, r22, r23, r24
....
r11: r*8: min x-coordinate
r12: r*8: max x-coordinate

r13: r*8: min y-coordinate
r14: r*8: max y-coordinate

This card is used in conjunction with *AUTM, or *STLF to define the rectangular fixtures on the bottom of the substrate. All nodes within the defined rectangles are fixed in all 3 axes. Figure 3.23 illustrates the option.

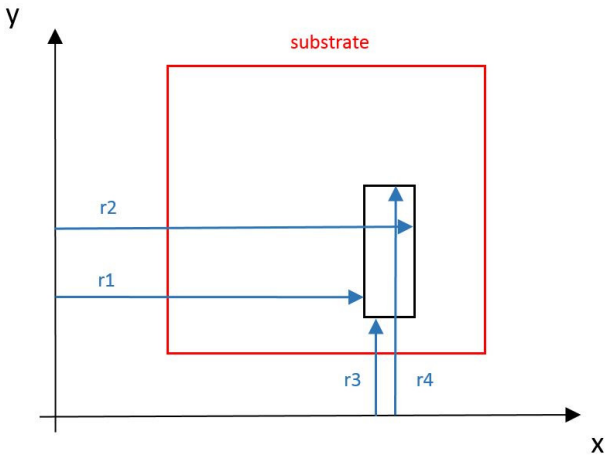


Figure 3.23: Positioning of a rectangular constraint

Best Practices:
Use *FIxR if practical to avoid overconstraining the part.

*FIxR: FIxture Rectangular

*FIxR
r11, r12, r13, r14, i11, i12, i13, i14
r21, r22, r23, r24, i21, i22, i23, i24
....
r11: r*8: min x-coordinate
r12: r*8: max x-coordinate
r13: r*8: min y-coordinate
r14: r*8: max y-coordinate
i11: i*4: x-coordinate [0 for free, 1 for fixed]
i12: i*4: y-coordinate [0 for free, 1 for fixed]
i13: i*4: z-coordinate [0 for free, 1 for fixed]
i14: i*4: load case number

This card is similar to *FIXR and *FIXC. However, it offers more flexibility to specify, which degree of freedom is fixed. It also allows specification of load case number.
Best Practices:
Using *FIxR allows for greater control than *FIXR and *FIXC and is less prone to overconstrain the part, which may produce erroneous results. **Ensure that when using *FIxR 6 degrees of freedom are restrained.** Fewer than 6 restrained DOF will allow bulk motion of the component during simulation while using more than 6 DOF will overconstrain the part, which may incur

erroneous simulations of distortion and stress. As of this version there is no warning given when under or over constraining the simulated part.

*FIxZ: FIxture rectangular with Z bounds

*FIxZ
r11, r12, r13, r14, r15, r16, i11, i12, i13, i14
r21, r22, r23, r24, r25, r26, i21, i22, i23, i24
....
r11: r*8: min x-coordinate
r12: r*8: max x-coordinate
r13: r*8: min y-coordinate
r14: r*8: max y-coordinate
r15: r*8: min z-coordinate
r17: r*8: max z-coordinate
i11: i*4: x-coordinate [0 for free, 1 for fixed]
i12: i*4: y-coordinate [0 for free, 1 for fixed]
i13: i*4: z-coordinate [0 for free, 1 for fixed]
i14: i*4: load case number

*FIxZ expands further upon the controls allowed by *FIxR, describing cubes wherein the specified directional displacements are fixed, as opposed to the area described by *FIxR and similar mechanical constraints which are applied to the base of the substrate.

*FISR: FIxture Spring Rectangular

*FISR
r11, r12, r13, r14, i11, i12, i13, i14, i15
r21, r22, r23, r24, i21, i22, i23, i24, i25
....
r11: r*8: min x-coordinate
r12: r*8: max x-coordinate
r13: r*8: min y-coordinate
r14: r*8: max y-coordinate
i11: i*4: x-coordinate [0 for free, 1 to add spring]
i12: i*4: y-coordinate [0 for free, 1 to add spring]
i13: i*4: z-coordinate [0 for free, 1 to add spring]
i14: i*4: material ID
i15: i*4: load case number

This card is similar to *FIxR. However, instead of fixing degrees of freedom, it adds nonlinear springs at specified degrees of freedom. The stiffness of the spring is defined using the *DASH card.

*FSRD: Fixture Spring Rectangular Death

*FSRD

When this option is used, the springs created with *FISR get removed when the substrate is removed from machine.
Required cards:*FISR

*SBBC: SuBstrate Boundary Conditions

*SBCC
i1

This card is used to automatically include boundary conditions for DED simulations. There are two supported options, i1 = 1, simply supported substrate, or i1 = 2, cantilevered substrate. These options are depicted in Figure 3.24. The simply supported substrate fixes nodes at the 3 corners of the model substrate, one in X,Y,Z, one in X and Z, and one only in Z. The cantilevered option fixes 2 corner nodes on just one longitudinal side of the build plate, both in X,Y, and Z.

*RBMO: Rigid Body Motion Optimization

*RBMO
i1, i2, r1, r2, r3

i1: Use 0 to disable optimization, or use any other value to enable (default).
i2: Maximum number of optimization iterations. Default 10000.
r1: Optimization tolerance. Default 1.d-10.
r2: Initial rigid body translation for optimization search as a multiple of the diagonal of the bounding box. Default 0.01.
r3: Initial rigid body rotation in radians for optimization search. Default 0.1.

By default, at the end of the mechanical part scale analysis an optimization process is completed to ensure that no artificial rotations or translations are introduced into the model results from the build plate removal simulation step. The Nelder-Mead optimization algorithm is used to minimize the RMS displacement, weighted by elemental volumes. A summary of the optimization process is listed in the log file.

3.9 Additional Options

*COOL: Additional COOLing step

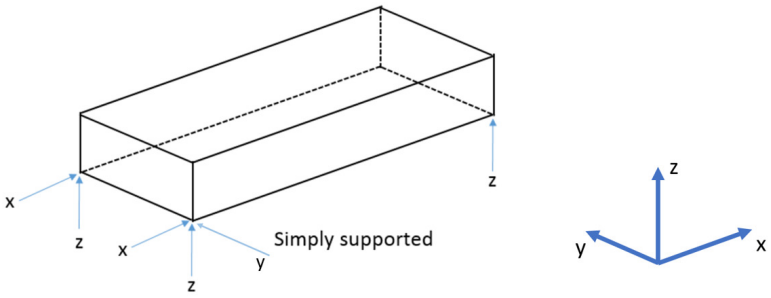
*COOL

This card adds one more time increment in mechanical analyses to simulate cooling back to the initial part temperature. This option is automatically turned on when using *PBPA.

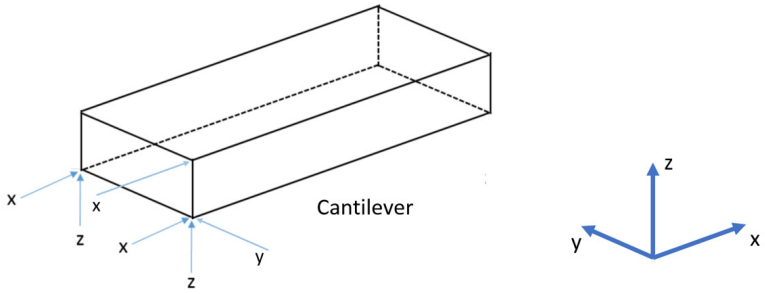
*AUTM: AUTo Mesh generation

*AUTM

This card automatically generates an input mesh for moving source analyses, including PRM generation and direct process simulations.



(a) *SBBC 1: Simply supported substrate



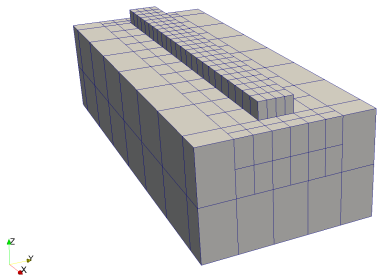
(b) *SBBC 2: Cantilevered substrate

Figure 3.24: Automatic DED Substrate Boundary conditions assigned by *SBBC

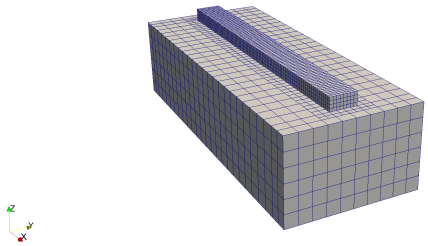
For moving source powder-bed analyses, *LSRP and *DDM! specify model volume and thickness, respectively, while the mesh parameters are determined by *ADAP and *ADP1.
For direct processes, geometry is defined by the deposition volume in *LSRF and depth set by *DDM! and the mesh refinement is controlled by by *ADAP, *NELR, and *ADP1.
Additional substrate volume can be specified by extension via *SBXY or by specifying minimum and maximum X and Y coordinates using *SBDM. Substrate depth is also controlled via *DDM!.

*NELR: Number of ELements per Radius

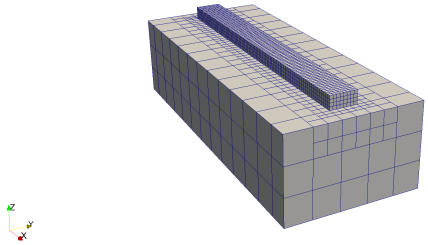
*NELR
r1
r1: r1*4: Number of element per melt pool radius. Default 1.
This card is used in conjunction with *AUTM and *ADAP to define the number of elements per melt pool radius in all three planes. Integer or real values are accepted.
Figure 3.25 illustrates the use of the *NELR card. Note that the change in elements per laser radius will scale up the total number of elements in the model. It is recommended to increase the *ADAP value to inhibit extraneous element production.



(a) NELR = 2, *ADAP = 2



(b) NELR = 4, *ADAP = 2



(c) NELR = 4, *ADAP = 3

Figure 3.25: The effect of *NELR upon the auto generated mesh.

*PARL: Parallel Activation Element Number

*PARL
i1

i1: number of elements in mesh above which parallel execution is turned on. Default 500

*SBXY: SuBstrate XY extensions

*SBXY
r1, r2, r3, r4
r1: r*8: x1-extension
r2: r*8: x2-extension
r3: r*8: y1-extension
r4: r*8: y2-extension

This card is used in conjunction with *AUTM, or *STLF to define an extension of the substrate beyond the bounding box of the part. Figure 3.26 illustrates the option.

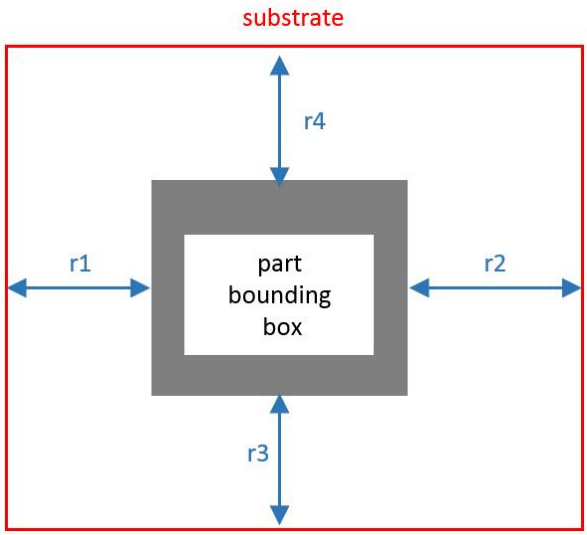


Figure 3.26: Assigning substrate geometry using *SBXY

*SBDM: SuBstrate DiMensions

*SBDM
r1, r2, r3, r4
r1: r*8: min x-coordinate
r2: r*8: max x-coordinate
r3: r*8: min y-coordinate
r4: r*8: max y-coordinate

This card is used in conjunction with *AUTM, or *STLF to define the XY coordinates of the substrate beyond the bounding box of the part. Figure 3.27 illustrates the option.

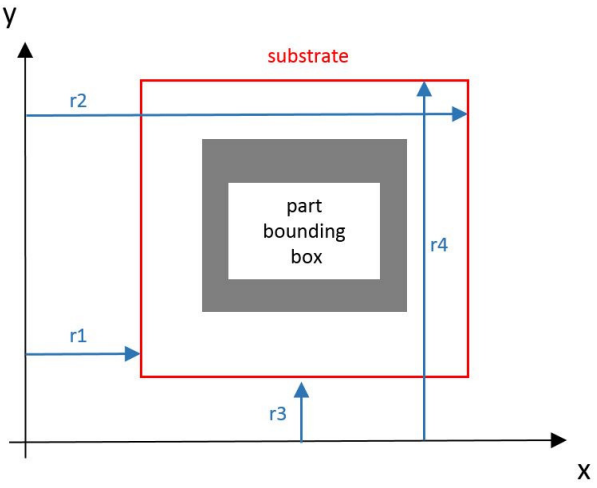


Figure 3.27: Assigning substrate geometry using *SBDM

*NELN: Maximum number of elements per node

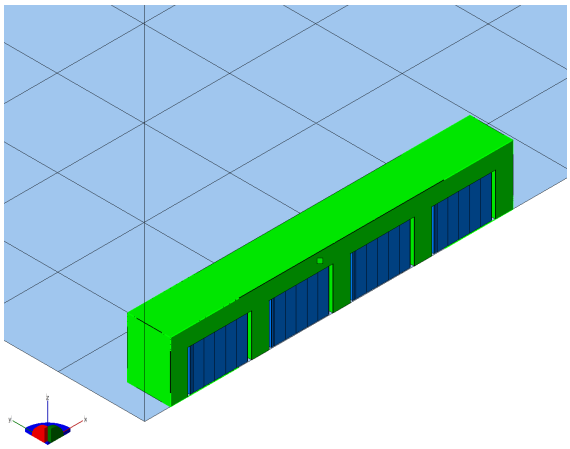
*NELN
i1
i1: i*4: Max number of elements per node. Default 20

This card is used to increase the max number of elements allowable per node in temporary pre-processing arrays. The card is needed in highly irregular meshes.

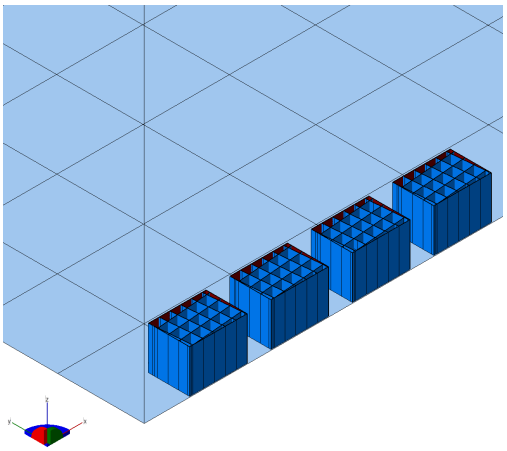
3.10 Support Structure Modeling

*CLIF: CLI File

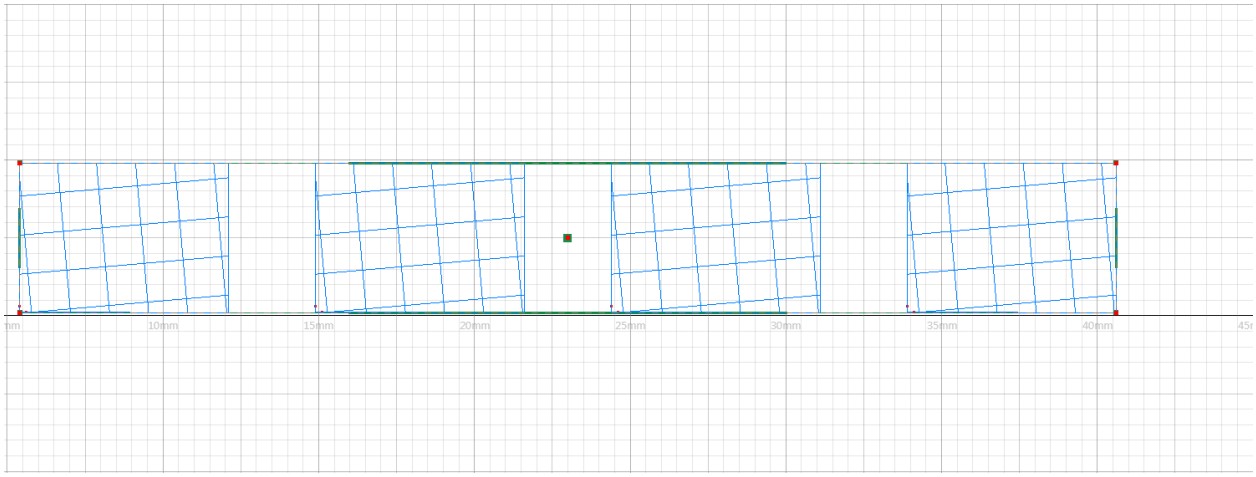
*CLIF
a1
a1: a80: Name of CLI file
This card inputs the support structure CLI into the model. This automatically adds an additional iteration at the end of the simulation which removes the support structure from the model mesh. If portions of the CLI model overlap with the STL model, Netfabb Simulation considers will consider the co-located volume as fully dense. This card must be applied in both the thermal and mechanical input files when using support structures in the model. At this time only box type supports can be used in the model. Figure 3.28 shows an example of compatible box type supports and an example layer slice of the supports. These can be easily generated using Netfabb® with the provided automated support script box_type_no_fragments.support
Best Practices: STL based supports are simpler to produce and have smaller file sizes which makes them preferable. It is suggested that *STLF and *STLM be used to import STL type support structures and assign material property IDs and volume fractions.



(a) Example box type supports on STL



(b) Example box type supports



(c) Example box type layer slice

Figure 3.28: Example box type suppports

*VCLI: Volume of CLI

*VCLI
r1
r1: Volume fraction threshold for CLI files
This card sets the volume fraction threshold for activation of CLI generated elements. The volume fraction is the ratio of the volume of the elements in the mesh to the volume fraction of the source CLI. Elements with volume fractions lower than the threshold are not activated. The default value is 0.05.
Required Cards: *CLIF

*UTSR: Ultimate Tensile StResss

*UTSR
r1
r1: Tensile stress of support material
This card is used to simulate support structure failure during powder bed processing. This card specifies the ultimate stress of the support material within the *MATE card block for the build material properties. When the von Mises stress of a Gauss point for an element at the support-build interface exceeds the specified ultimate tensile stress, the element fails, producing a warning in the output file when the failure occurs and a warning at the end of the simulation stating what percentage of the total support contact volume has failed. A failed element remains in the mesh but does not resist deformation. An example of failed elements is shown in Figure 3.29.

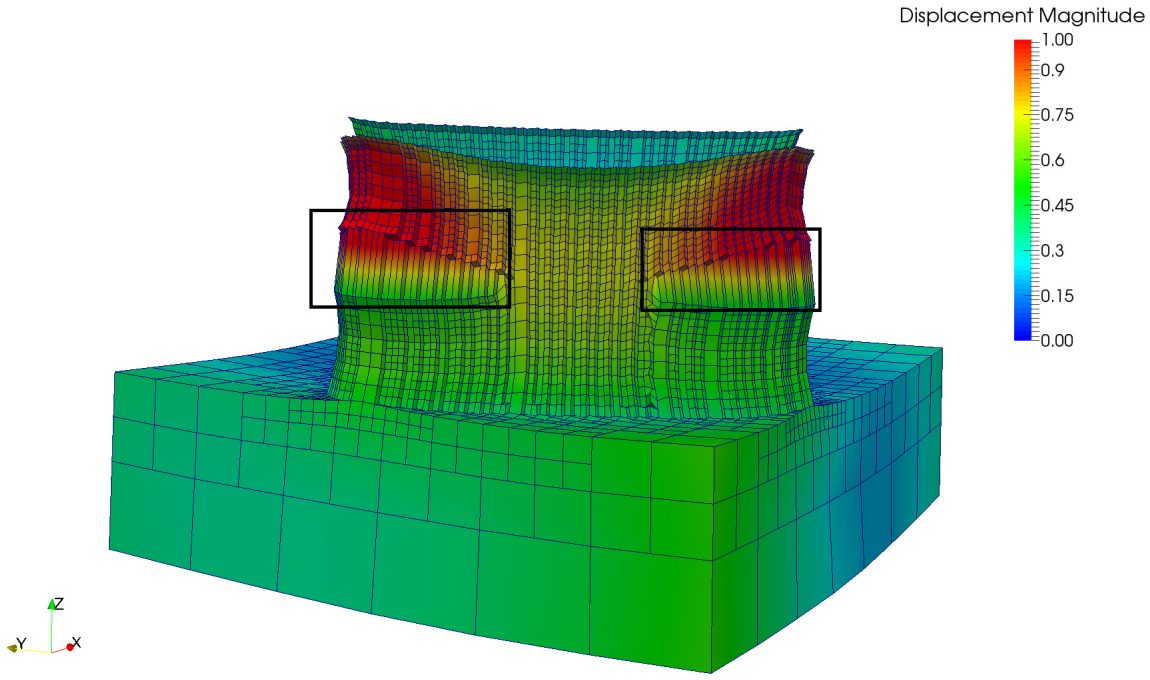


Figure 3.29: Example of failed elements using *UTSR, 5X magnification

3.11 Heat treatment modeling

Heat treatment modeling is a beta feature. Heat treatment must be used in conjunction with part level plasticity via the *PPLA card. To simulate heat treatment two cards are required, one to set the heat treatment schedule *STRF and one to set the annealing temperature for one or more materials *STRR. Heat treatment can be modeled before or after either build plate or support structure removal and in any combination thereof using *HTOR and *HTAT, along with the additional heat treatment schedule(s) required via *STR1 and *STR2

*STRF: STress RelieF heat treatment simulation

*STRF
r11, r12: temperature 1 [°C], time 1 [s]
r21, r22: temperature 2 [°C], time 2 [s]
r31, r32: temperature 3 [°C], time 3 [s]
...

This card is used to set the temperatures and times for stress relief heat treatment. Heat treatment modeling requires plasticity to be enabled by addition of the *PPLA card. Stress relaxation (recrystallization) temperatures must be specified using the *STRR card to enable stress relief simulation. Cooling rates can be controlled using *ACNV
The times in the *STRF card refers to time after the end of the thermo-mechanical build simulation and post process cool down. The heat treatment modeling occurs after the plasticity modeling but before the part is removed from the build plate. If a final temperature has been specified by *FINT, that is the starting temperature for the heat treatment simulation. If no final temperature is specified, then the initial temperature set by *INIT is used. If an initial temperature has not been explicitly stated, then the ambient temperature is used as the heat treatment starting temperature, as set by *AMBI. Additional time steps may be added based upon convergence requirements.
Required Cards: *STRR, *PPLA
Best practices: For heat treatment where the final cool down takes an unknown amount of time, it is recommend a long period of time [12 hours+ depending on the thickness of the part] be added between the second to last and last heat treatment step to ensure the part may naturally come to the specified final temperature. Follow the meshing practices advised under the *PPLA best practices section of using *PBLR values of 0 or 1, and performing additional mesh convergence checks to ensure the mesh is dense enough to adequately capture the modeled behavior.

*STRR: STRess Relaxation temperature

*STRR
r1 : Stress relaxation or recrystallization temperature.
This card is used in conjunction with *STRF inside the material block or blocks to specify the temperature(s) at which stress is relaxed during heat treatment simulation. An *STRR card must be included for each specified material via *MATE and *MATI. For non-homogenous build and build plate materials, plasticity properties must be included in the build plate material block, as also stated by *PPLA.
Required Cards: *STRF, *PPLA
Best practices: For materials with unknown stress relaxation or recrystallization temperatures, use a value between 0.3-0.5 of melting temperature. No stresses will be relaxed if the *STRR value

is higher than the peak temperatures specified by the heat treatment in *STRF.

*ACNV: Annealing CoNVection

*ACNV
r11, r12: Convection value 1, temperature 1
r21, r22: Convection value 2, temperature 2
...

This card is used to control the temperature dependent convection of the heat treatment process. By default the value is 0.000010 $W/mm^2/K$.
Required Cards: *STRF, *PPLA, *STRR

*HTOR: Heat Treatment ORder

*HTOR
i1: Allowable values of 0-2
i1 = 0 Heat treatment before build plate and support removal
i2 = 1 Heat treatment after build plate removal but before support removal
i2 = 2 Heat treatment after both build plate and support structure removal

This card is used to determine the order in which heat treatment occurs. By default, if the *HTOR card is not used, heat treatment occurs after cooldown and plasticity, but before build plate and support removal. This uses the heat treatment schedule specified by *STRF.
Required Cards: *STRF, *PPLA, *STRR

*HTAT: Heat Treatment AnyTime

*HTAT
i1, i2, i3: Logical binary 0 or 1
i1 = 1 Model heat treatment before build plate and support removal
i2 = 1 Model heat treatment after build plate removal but before support removal
i3 = 1 Model treatment after both build plate and support structure removal

This card is used to allow users to specify multiple heat treatment events during a simulation. A value of 0 indicates no heat treatment at that stage, 1 indicates heat treatment should occur at that stage. When both *HTAT and *HTOR are specified, *HTAT takes precedence and *HTOR is ignored. All heat treatment stages use the same stress relaxation value set by *STRR.
Note: The heat treatment schedule for the case when i1 = 1 is specified via *STRF. The heat treatment schedule for the case when i2 = 1 can specified via *STR1. The heat treatment schedule for the case when i3 = 1 can specified via *STR2. In case *STR1 is not included in the input deck, it is assumed that the heat treatment schedule is the same for all three stages of the heat treatment simulation, and the heat treatment schedule is specified via *STRF
Required Cards: *STRF, *PPLA, *STRR

*STR1: STress RelieF heat treatment simulation 1

*STR1
r11, r12: temperature 1 [°C], time 1 [s]

r21, r22: temperature 2 [°C], time 2 [s]
r31, r32: temperature 3 [°C], time 3 [s]
...

*STR1 is used to specify an additional heat treatment schedule when *HTAT i2 = 1, heat treatment between build plate removal and support structure removal.
Required Cards: *HTAT, *PPLA, *STRR

*STR2: STress RelieF heat treatment simulation 2

*STR2
r11, r12: temperature 1 [°C], time 1 [s]
r21, r22: temperature 2 [°C], time 2 [s]
r31, r32: temperature 3 [°C], time 3 [s]
...

*STR1 is used to specify an additional heat treatment schedule when *HTAT i3 = 1, heat treatment after both build plate and support structure removal.
Required Cards: *HTAT, *PPLA, *STRR

Chapter 4

Post processing tools

Included with the solver are programs for post process analysis along with warped or compensated STL files.

timex: Nodal results program

`timex` is a command line program that allows the user to interogate the completed results at specified points to get temperature histories from thermal simulations or stress, strain, and distortion histories from mechanical results. For points not associated with model nodes, values are linearly interpolated. This program works for both moving source and part scale simulations. The usage of this program is:

```
timex input-example.txt
```

The input file, `input-example.txt` is a text file of the following format:
***INPU : Netfabb simulation input file name, without .in extension**
a1: Input file name

***PNTS : Number of points and point coordinates**
i1 : Number of Points
r11, r12, r13 : X1, Y1, Z1 coordinates
r21, r22, r23 : X2, Y2, Z2 coordinates
...

***RESU** i1
: number of result components. 1 for scalar, 3 for vector, or 6 for symmetric tensor. a1
a1: result filename extension. See the table below.
By default, without using *RESU, timex run on a thermal analysis will return the temperature history, while a mechanical file will return the displacement history. To override this, specify tensor order i1 and the file extension a1.
==========
Result card — file extension — description(tensor order)

```
============================================================
OSIG — .sig.ens — Cauchy stress (6)
OEPT — .ept.ens — Elastic strain(6)
OEQP — .eqp.ens — Equivalent plastic strain(1)
OMLT — .mlt.ens — Melt indicator(1)
OEPP — .epp.ens — Plastic strain(6)
OSD3 — .sd[123].ens — Principal stress directions(3)
OSP3 — .sp[123].ens — Principal stress values(3)
OSVM — .svm.ens — von Mises stress(1)
q@"J — .tmp.ens — Temperature(1)
============================================================
```

For a default case example, having run a simulation using the file `test_input.in` the user extracts the temperature or stress and distortion history at 3 points in a vertical column:

```
*INPU
test_input
*PNTS
3
5, 5, 0
5, 5, 2
5, 5, 4
```

To extract the Cauchy Stress Tensor at the same location, the timex input file would be:

```
*INPU
test_input
*PNTS
3
5, 5, 0
5, 5, 2
5, 5, 4
*RESU
6
sig.ens
```

Additional options for multi-layer adaptivity
For Multilayer adaptivity simulations, two additional cards need to be added to the timex input file, ***CRSE** and ***SHFT**.
***CRSE: CoaRSE case file**
This card tells timex to access the coarse instead of fine case file.
***SHFT: SHiFT to multilayer adaptivity** This card tells timex to shift to the multilayer adaptivity node inquisition scheme.

distort_stl

`distort_stl` is a command line program that is used to produce warped or compensated STL file geometries from a thermo-mechanical part scale simulation. Warped geometries can be used for part validation or to check if the distorted part will fit within tolerance in an model assembly. Compensated geometries alter the source STL using calculated displacements to correct for the

predicted distortion, leading to a less distorted part when the compensated geometry is built in a powder bed machine. This function can also produce warped STLs from solid support structures. The usage for this program is:
`distort_stl input.txt`
Where the input file has the format:
***INPU : Netfabb simulation input file name, without .in extension**
a1 : input file name

***WARP : Warpage magnification factor**
r1: magnification factor

This is the magnification factor which applies the modeled displacements to the source STL file. The default value -1 which produces a fully compensated STL file. A value of 1 will give warped displacements to be used for post processing analysis or assembly fit checks.
Best practices: Only produce compensated STLs for production after the material properties and processing parameters for the intended build have been experimentally validated. Unless an iterative simulation-compensation process is executed by using the ***REFG** card to converge upon a compensated model, values other than -1 for compensation may be used to calibrate the compensation algorithm. This is due to the non-linearity of the compensation problem which is not taken into account when performing a single simulation-compensation step. On a small part with measurable distortion print parts using values of -0.5 -0.75 and -1.0 to test the compensation efficacy.
***IOFF : Increment offset from end of analysis**
i1: increment offset
This controls the increment which the STL file is produced from. The default value is -2. Range [-<ninc>, -1] or [1, <ninc>], where <ninc> is the number of solver increments with results. Negative *IOFF counts backwards from the end of the analysis, and positive *IOFF counts forwards from the beginning.
Best practices: The preferred increment for compensation is the increment after cool down, but before build plate release or removal. If compensation or warping is to be performed using geometries after this step it is advised that plasticity should be turned on using *PPLA. If the part is to be heat treated, compensation should be performed after the heat treatment simulation process, before the removal of the part from the build plate. Consult *STRF for details about heat treatment simulation.
Additional optional controls are:
***OUTP : Output STL file name(s)** a1
output STL file name 1 a2
output STL file name 1 ...

By default the output name(s) will be <*INPU>_<i>_warp.stl, where <i> is indexed for each output STL according to *ASTL or *ISTL. Each *OUTP value must be unique, and cannot conflict with the input STL names under *STLF in the solver input file. If both *OUTP and *ISTL are used, then they must have the same number of arguments.
***NINT : Number of nodes for inverse distance weighted nearest neighbor interpolation**
i1: number of nodes
Default value 10. Range [1, <nnode>), where <nnode> is the number of nodes in the solver mesh.

***EINT : Exponent for inverse distance weighted nearest neighbor interpolation**
r1: exponent
Default value 1.0. Range (0, ∞)

***SNAP : Snapping tolerance**
r1: snapping tolerance
Default value 1e-5. Range (0, ∞)

***MAXR : Maximum number of refinement levels**
i1: maximum refinement levels
Default value 10. Range [1, ∞)

***MINE : Minimum edge length**
r1: min length
Default value 0.5. Range (0, ∞)

***MAXE : Maximum edge length**
r1: max length
Default value 3.0. Range [<*MINE>, ∞)

***RTOL : Refinement tolerance**
Default value 1e-3. Range [0, ∞)

***FORM : Output STL file format**
a1: a*1: output format, Either "a" or "b" where "a" = ascii format and "b" = binary format.
Default value "b".

***ISTL : STL ID(s) from list of part STLs in *STLF**
i1
i2
i3
...
i1: STL ID
Default value 1. Range [1, <nstl>], where <nstl> is the total number of part STL files listed under *STLF in the solver input file. Support STLs do not count towards <nstl> unless the *SSTL card is used.

***ASTL: output All STLs**
Default off. Instead of using *ISTL to explicitly list each STL to be warped and exported, *ASTL can be used to export all STLs. If both *ISTL and *ASTL are used, then *ISTL is ignored.

***SSTL: output Support STLs**
Default off. Including this card will export warped STLs for support structures. This is intended to be used only for volumetric support files, not zero thickness or pre-homogenized STL files. Individual files can be exported by using *ISTL or all files, support and part, can be exported using *ASTL.

***O3MF: Output 3MF file**
Default off. Use this card to export warped 3MF files in addition to STL files. Note each geometry will be written to its own STL file.

***O3DM: natively export warped geometry to colored 3D Manufacturing Format (3MF) file(s)** Default on. This card enables the output to 3MF files with the same base filename as the STL files and a "color.3mf" filename extension. This option natively exports the data from distort_stl which keeps the connectivity intact. Displacement magnitudes are encoded in the 3MF as a blue-to-red rainbow colormap.
***N3DM: turn off colored 3DF export (*O3DM)** Default off. Use this to override *O3DM.
***WRTU: pointcloud export**
Default off. The *WRTU card will create a *_warp_wrtu.txt file. Columns 1: 3 are nominal position x, y, z for all refined STL vertices and columns 4: 6 are displacement x, y, z. Consult the solver input card entry of the same name *WRTU for additional details.
***FIXZ: fix the z coordinates of vertices on the bottom of the part**
Default off. The *FIXZ card keeps the bottom of the compensated geometry flat to maintain contact with the build plate.
***ZTOL: z tolerance for *FIXZ**
r1: z tolerance
Default r1 = 1.d-5. Vertices within r1 of the minimum z coordinate are fixed.
***REFG: reference geometry**
a1
a2
a3
...
ai: reference geometry STL file name
Default null. Currently only 1 reference STL is supported. The *REFG card specifies the reference or nominal geometry to be used for iterative compensation, as opposed to the preform geometry (from the solver *STLF card) or the compensated geometry (output of distort_stl).
***STLF: STLFile** **Tech Preview** a1: Reference DED Geometry STL file name
Using this card allows for the compensation of DED type simulations by referring to an STL file depicting the nominal geometry deposited during simulation.

line_plot

`line_plot` is a command line program used to produce 2d plots of simulation results. The usage of this program is:
`line_plot input-example.txt`
Where the input file is a text file of the following format:
***INPU : Netfabb simulation input file name, without .in extension**
a1: Input file name

***IPLT : Increment offset from end of analysis**
i1: increment offset
This controls the increment which the STL file is produced from. The default value is -2. Range [-<ninc>, -1] or [1, <ninc>], where <ninc> is the number of solver increments with results. Negative *IPLT counts backwards from the end of the analysis, and positive *IPLT counts forwards from the beginning.
***LXYZ: Line direction and coordinates**
a1
r1,r2

a1: Cartesian line direction, can be x, y, or z.
r1: First line coordinate
r2: Second line coordinate

The line direction determines the remaining two coordinates set by r1 and r2, so that:
a1 = x, r1 = y coordinate, r2 = z coordinate
a1 = y, r1 = x coordinate, r2 = z coordinate
a1 = z, r1 = x coordinate, r2 = y coordinate

***CRSE: Use coarse results input files** *CRSE
This card uses the coarse *_c.case and *_c.geo files for analysis.
***RESU: Number of result components and result file extension** *RESU i1
a1

i1: Number of result components, may be 1 for scalar, 3 for vector, or 6 for symmetric tensor. a1: Result file extension name.
Default: i1 = 1, a1 = .tmp.ens for thermal model results or a1 = .dis.ens for mechanical models.
***OUTP: Output file name** a1

a1: Output file name. By default the output name will be `line_plot_inputfilename.csv`

prm_scale

`prm_scale` is a command line tool used to create a new PRM file from an existing one by applying a corrective scaling factor. This should be used when simulation results consistently over or under predict experimentally measured distortion. Usage:
`prm_scale a1 a2 r2`

a1: Input prm file name
a2: Output prm file name
r2: Scaling factor

For example, assume through repeated prints and measurements it is found that the generic Ti-6Al-4V prm file is 23% high due to using slightly different processing conditions than the processing parameter was generated with, using a pulsed laser system, etc. The file would be scaled thusly:

```
prm_scale Ti-6Al-4V_generic.prm Ti-6Al-4V-Scaled.prm 0.77
```

Chapter 5

Units

While essentially unitless, Netfabb Simulation has been written to follow the mm/kg/J/s standard for units. The user may use an equivalent system of units after ensuring the relations are consistent. A brief list of the primary input and output variables is given with their associated de facto units.

Angle ° (Degrees)

Convection W/(mm^2 °C)

Density kg/mm^3

Displacement mm

Elastic (Young's) Modulus N/mm^2

Hatch spacing mm

Laser absorptivity coefficient W/W (unitless)

Laser radius mm

Laser power W

Latent heat J/kg

Position mm

Specific heat capacity J/(kg °C)

Strain mm/mm (unitless)

Temperature °C

Thermal conductivity W/(mm °C)

Thermal emissivity W/W (unitless)

Thermal expansion coefficient μm /(μm °C) (or 1/°C)

time s

Velocity mm/s

Yield strength MPa

Chapter 6

Common errors and warnings

This chapter describes common errors, critical warnings, and warnings the user may experience using Netfabb Simulation .
Errors: any error will stop the simulation, indicating that there was a fatal flaw in the setup or execution of the simulation.
Critical Warnings: Critical warnings will allow the simulation to proceed, but indicate that a simulation setting was not valid and will be overridden. Depending on the severity of the change, users may want to terminate the simulation manually, adjust model settings, and re-run the simulation.
Warnings: Warnings alert the user when some some settings are ignored, or overwritten, in addition to recording support structure failure and recoater blade interference. Warnings should not require the simulation to be rerun.
Errors will stop the simulation, while warnings will alert the user but continue to let Netfabb Simulation run.
String variables are denoted using ??, e.g. filename1??.in
Numeric variables are denoted using ##, e.g. laser-line##

6.1 Errors

6.1.1 Autodesk Netfabb Local Simulation LT Errors

```
Number of coarsening generations set by *PBLR exceeds max limit 1.  Consider
upgrading to Netfabb Local Simulation.
Number of layers grouped per element *PBPA exceeds max limit 20.  Consider
upgrading to Netfabb Local Simulation.
```

For users with the Netfabb Ultimate simulation product, Autodesk Netfabb Local Simulation LT, *PBLR is capped to a maximum of 1 and *PBPA is capped to a maximum of 20. If these are exceeded in the input files these errors will be displayed. Change the mesh settings so that they are within the acceptable limits and rerun the simulation.

6.1.2 Runtime errors

```
Array allocation error.  This may be caused by insufficient memory.  Close other
memory intensive programs or run with a less dense mesh.  Allocation error number
= ##
```

`Not enough memory to run Pardiso solver. Additional RAM required to run in-core: ## MB. Additional RAM required to run out-of-core: ## MB. Close other RAM intensive programs and rerun using out core options *OFC2 and *OFC3, with reduced mesh density, or on a system with more RAM`
These errors indicate that the workstation being used does not have enough RAM to run the Netfabb Simulation model. Install more RAM or run on a higher memory system. The suggested minimum system requirements are 14 physical processing cores and 64 GB of RAM.
`fort severe -1`
This error indicates a segmentation fault and that the user has run into an unresolved issue in the software. Please immediately email the input files required to reproduce the error to Netfabb Simulation customer support at: projectpansuppport@autodesk.com

6.1.3 Licensing errors

`Netfabb Simulation has not been able to acquire a license to run, please contact Autodesk`

These errors occur when there is an error in the license file. This could be due to an expired license, a corrupted license file, or pointing at the wrong license file. Ensure an active license file is in the location specified during installation. If problems persist, try reinstalling Netfabb Simulation , restart the license server from LMTOOLS, or contact customer support.
Note: For Node Locked licenses only one simulation can be run at a time. If a second instance of Netfabb Simulation is initialized, it will return one of the above errors. If you are unsure if Netfabb Simulation is already running:
Windows Install:
1. Open the task manager by pressing `CTRL+SHIFT+ESC`
2. Click on the `Processes` tab
3. Look for `pan.exe` in the `Image Name` column
4. Click the `End Process` button if needed

Linux Install:
1. In the terminal type `top` to obtain a list of currently running processes.
2. Look for `pan`.
3. If needed, end the process using the `kill PID` command, where `PID` is the process number of the pan instance in the top listing, under the column heading `PID`

6.1.4 Netfabb Simulation LT errors

Netfabb Simulation LT, included with the Netfabb Ultimate package, has limitations regarding the type and complexity of the simulations that may be completed using this software.
`Moving source models are not available with Netfabb Simulation LT. Consider upgrading to Autodesk Netfabb Simulation.`
Moving source models, either PRM generation or DED simulations, are not accessible using the LT product.
`Layer-nodes exceed max limit 20M. This model exceeds the complexity allowed in Local Simulation LT. Coarsen the mesh or upgrade to Local Simulation.`
`Nodes exceed max limit 500000. Coarsen the mesh or upgrade to Local Simulation.`

These errors occur when the settings create a mesh that exceeds the limitations of the LT product. Increase the *PBLR and *PBPA values, if possible. If loose powder *+PDR is enabled, consider using convection boundary conditions instead via *CONV to reduce the mesh demands.
`Full analysis is not available with Netfabb Premium. Consider upgrading to Netfabb Ultimate or Netfabb Local Simulation.`
Netfabb Premium is only capable of performing mesh previews.

6.1.5 Basic input file errors

`Cannot read file filename??.in. The input file has incorrect formatting or has been corrupted.`
`Cannot open file filename??.in . Check spelling and ensure the file is in the correct directory.`
These errors indicate the input file specified by the user does not exist. Check for spelling mistakes and that Netfabb Simulation has been executed in the correct directory.
`File filename?? does not exist. Check the input files are correct and ensure filename is in the correct directory.`
This error message occurs when the specified input, PRM, or STL file cannot be found in the current directory.
`Command-line argument *???? was unable to be read. Check input files and ensure usage is accordance with the user manual.`
`Cannot read file filename due to error argument in card: *????. Refer to the user manual for correct spelling and usage.`
`Cannot read file filename??.in due to error in card: *????. Refer to the user manual for correct spelling and usage.`
These error messages are displayed when an unknown line is read from the input file. This is commonly due to accidentally uncommenting a comment line, typing errors, or using a non-basic text (ANSI or ASCII) format.
`Command-line argument ???? was truncated. Correct in the input file and rerun.`
This error message indicates one of the control cards has been truncated. Fix the input file and rerun.
`Could not read dependent analysis files. The file may be missing, the wrong file type, or corrupted.`
This occurs when one or more files required to run a simulation are missing. For a mechanical analysis, ensure the correct dependent thermal analysis is listed under *DEPE and that the thermal analysis completed correctly.
`End of input file filename??.in. One or more input cards are incorrectly formatted. Check each card has the correct usage specified by the user manual.`
This error indicates the input file read to the end of the file without encountering in the *END card, or with one or more formatting errors in prior control cards.
`Could not read restart file example.bin. Check the restart file name specified by *REST.`
This error indicates that the .bin file specified by the *REST card is not a valid restart file and may have been corrupted.

PRM file read errors

`PRM file example??.prm is not correctly formatted. Ensure the correct file is being used. The PRM file may have been corrupted and needs to be regenerated.`
`Cannot read prm file example??.prm. Ensure the correct file is being used. The PRM file may have been corrupted and needs to be regenerated.`
These errors occur when the PRM file in *PBPF is misnamed, corrupt, or not a prm file. Locate the correct file or regenerate the PRM file.
`No PRM file specified by the PRM file card *PBPF. Specify at least 1 PRM file under the *PBPF card.`
This error is shown when the *PBPF card is included, but without any PRM files listed.
`This PRM file was created by pan2 and is not valid. Regenerate the PRM file using prm_gen.`
PRM files must be generated with the prm_gen command, not with the pan or pan2 command. Regenerate the PRM file using the prm_gen command. Refer to Example 1 in the examples manual or in this manual in the prmgen section.
`Zero entries found in example.lsr. Check the .lsr file or the *LSRP settings if the file was auto-generated.`
This error indicates the .lsr file generated during PRM generation is empty. Correct the settings in the *LSRP card and regenerate.
`The number of ambient temperatures are not consistent in the PRM files. Ensure the correct PRM files are being used. You may need to regenerate the PRM files to ensure the specified temperatures are identical in each file.`
`The number of lack of fusion temperatures are not consistent in the PRM files. Ensure the correct PRM files are being used. You may need to regenerate the PRM files to ensure the specified temperatures are identical in each file.`
`The number of hot spot temperatures are not consistent in the PRM files. Ensure the correct PRM files are being used. You may need to regenerate the PRM files to ensure the specified temperatures are identical in each file.`
`The interlayer temperatures are not consistent in the PRM files. Ensure the correct PRM files are being used. You may need to regenerate the PRM files to ensure the specified temperatures are identical in each file.`
`The lack of fusion temperatures are not consistent in the PRM files. Ensure the correct PRM files are being used. You may need to regenerate the PRM files to ensure the specified temperatures are identical in each file.`
`The hot spot temperatures are not consistent in the PRM files. Ensure the correct PRM files are being used. You may need to regenerate the PRM files to ensure the specified temperatures are identical in each file.`
If multiple PRM files with lack of fusion or hotspot data are used and they have not been generated using the identical number and values for interlayer temperatures, lack of fusion temperatures, and hotspot temperatures, one or more of the above errors will be displayed. Check that the correct PRM files have been specified and regenerate any PRM files as necessary.
`File example??.prm does not contain any parameter or lack of fusion entries. Ensure the correct file is being used. The PRM file may have been corrupted and needs to be regenerated.`
`File example??.prm does not contain any parameter entries. Ensure the correct file is being used. The PRM file may have been corrupted and needs to be regenerated.`

These errors indicate that the PRM file does not contain standard PRM data. This may be due to the PRM generation process not completing, the file being corrupted, or accidentally using a non PRM file.

```
Number of prm files exceeds 100.  Increase the value of maximum number of
materials set by *NMAT.
```

By default a maximum of 100 PRM files may be entered. Use *NMAT to increase this limit.

6.1.6 Thermal-mechanical input file mismatch errors

```
The dependent mesh is not the same.  Check that mesh settings *PBPA and *PBLR for
part analyses or *ADAP, *ADP1, and *NELR for moving source analysis, are
identical between the thermal and mechanical input files.
```

This error occurs when different mesh settings (or Patran meshes) are used for the thermal and mechanical simulations resulting in different numbers of elements in the two meshes. Ensure all mesh controls are identical in the two input files.

```
Could not read dependent thermal binary results example.bin.  The thermal
analysis may have not been completed.  Check the thermal .out log file and rerun
if necessary.
```

This error will be shown if the .bin file specified by a mechanical analysis does not exist, is incomplete, or corrupted. Ensure the correct dependent file is set by the *DEPE card. Check the log files to ensure the thermal analysis completed successfully.

6.1.7 PRM generation errors

```
Number of lack of fusion temperatures in file example??.prm is different than
that specified in command line.  Rerun with same number the lack of fusion
temperatures or save to a new PRM file.
Lack of fusion temperatures in file example??.prm are different than that
specified in command line.  Rerun with the same lack of fusion temperatures or
save to a new PRM file.
Number of hotspot temperatures in file example??.prm is different than that
specified in command line.  Rerun with the same number of hotspot temperatures or
save to a new PRM file.
Hotspot temperatures in file example??.prm are different than that specified in
command line.  Rerun with the same hotspot temperatures or save to a new PRM
file.
```

These errors occur when a user attempts to append an existing PRM file which contains lack of fusion or hotspot data, but without matching the original temperature values. The user should either create a new PRM file or correct the temperatures. Consult prmgen for correct usage.

```
Power in file example??.prm is different than that specified in the input file.
Check processing parameters or save to a new PRM file.
Radius in file example??.prm is different than that specified in the input file.
Check processing parameters or save to a new PRM file.
Speed in file example??.prm is different than that specified in the input file.
Check processing parameters or save to a new PRM file.
Layer thickness in file example??.prm is different than that specified in the
input file.  Check processing parameters or save to a new PRM file.
Hatch spacing in file example??.prm is different than that specified in the input
```

`file. Check processing parameters or save to a new PRM file.`
`Interlayer dwell in file example??.prm is different than that specified in the input file. Check processing parameters or save to a new PRM file.`
These errors occur when a user attempts to append an existing PRM file but without matching the original processing parameters. The user should either create a new PRM file or correct the processing parameters. Consult prmgen for correct usage.
`Zero or negative activation temperature in the PRM file. Ensure that thermal analysis file has been run first, *GTAB is included, and that the file name specified by *GTAB is identical for both the thermal and mechanical input files.`
This error occurs when the mechanical PRM generation input file either cannot find the PRM file specified by the *GTAB card or the thermal results have not been written to the PRM file.
`Hatch spacing must be positive.`
`Diameter must be positive.`
`Travel speed must be positive.`
`Layer thickness must be positive.`
`Hatch spacing must be positive.`
`Dwell time must be non-negative.`
`Number of layers must be positive.`
`X width must be positive.`
`Y width must be positive.`

These errors occur when any of the above *LSRP values are negative.
`Argument 3: invalid power ##`
`Argument 4: invalid radius ##`
`Argument 5: invalid speed ##`
`Argument 6: invalid hatch dwell ##`
`Argument 7: invalid layer dwell ##`

These errors will be shown when the chosen *LSRP parameters are invalid.
`PRM dwell time multiplier *TPRM must not be less than 1.0. Increase the *TPRM value.`
This error occurs when the user inputs a PRM generation time scaling value, *TPRM, less than 1.
`PRM generation set by *GTAB cannot be used with substrate extension using either *SBXY or *SBDM. Remove the *SBXY or *SBDM cards from the PRM generation input files.`
PRM generation is not compatible with the substrate extension cards. Remove *SBXY or *SBDM and rerun prmgen
`Input file example??.in does not contain *LSRP required to generate a PRM file. Ensure the input files are correct and re-run prm_gen.`
PRM generation simulations requires *LSRP to specify the power, velocity, and basic scan strategy information used during processing.

6.1.8 Part scale geometry errors

`No STL file found. At least 1 stl file must be listed under the *STLF card.`
At least one STL file must be used in the *STLF card.
`STL file input *STLF is attempted with patran neutral file *INPU. Choose only one geometry input method.`

This error occurs when the user attempts to use both an .stl file and a Patran generated mesh simultaneously.

```
STL file input *STLF is used with no adaptivity card *ADAP. Add the *ADAP card to
the input files using proper values.
STL file import *STLF is used with no layer mesh grouping *PBPA. Add the *PBPA
card using proper values.
```

These errors indicate the proper meshing options have not been turned on using when using *STLF.

```
Min z coordinate in STL file is not attached to the substrate.  Use the *DDM!
card to adjust the substrate coordinates.
```

This error is displayed with the substrate top value specified by *DDM! is lower than the minimum z value in the part geometry. Check units of the STL file, alter the DDM! settings, or translate the imported geometry files.

```
Negative or zero number of layers.  Check if entire STL is inside substrate.
```

This error generally occurs if the top of the substrate set by the *DDM! card exceeds the maximum Z coordinate of the STL file. Fix the *DDM! values or regenerate the STL file at a different set of coordinates.

```
Netfabb repair was not successful.  Try to adjust repair settings with *NTFT, try
an extended repair with *NTFE, or repair in Netfabb.

One or more geometries could not be repaired.  Try to adjust repair settings with
*NTFT, try an extended repair with *NTFE, or repair in Netfabb.
```

When STL files with non manifold edges or other errors are detected by the solver, it attempts to repair the STL so that it can be meshed. This uses the Netfabb default repair script. Upon failing, it reattempts repair using the extended Netfabb Repair script. If this operation does not return a meshable STL, it produces one of the the above warnings indicating that more in depth STL repair will needed before the geometry file may be simulated.

```
Number of STL vertices must be positive.  Re-export the STL from the source
modeling software.
Number of STL triangles must be positive.  Re-export the STL from the source
modeling software.
STL padding tolerance *STOL must be non-negative.  Increase STL padding tolerance
to a minimum of 0.
```

This error occurs when the user has accidently input a negative value for the multiple STL tolerance value, *STOL.

```
The number of STL files specified by *STLF STL file import and *STLM STL file
mapping must be identical.
```

This error occurs when the user has a mismatch between the number of STLs imported by *STLF and the number of items in the STL mapping card *STLM.

```
Configuration for STL ID ## must be in range [1, 3].  Fix values in STL mapping
*STLM.
```

This error occurs when the user attempts to specify a configuration ID lower than 1 or higher than 3 in *STLM.

```
Configuration 2 for STL ID ## (substrate) is not supported yet.  Fix values in
```

```
STL mapping *STLM.
```

This error occurs when a user attempts to use the Configuration ID = 2 in *STLM.

```
PRM ID for STL ID ## must be in range [1, Number of STLS].Fix values in STL
mapping *STLM.
Material ID for STL ID ## must be in range [1, ## Number of STLs].  Fix values in
STL mapping *STLM.
```

These errors occur when the PRM or STL id set in *STLM is set higher than the number of prm files imported by *PBPF or the number of STL files imported by *STLF respectively.

```
Volume fraction for STL ID ## must be in range [0.0, 1.0].  Fix values in STL
mapping *STLM.
```

This error indicates that a volume fraction outside of the accepted bounds has been specified in the *STLM card.

```
The number of STL files specified by *STLF STL File and *STLC STL volume fraction
must be identical.
```

The *STLC, *STLF, and *STLM cards must all have the number of STL files specified.

```
The STL index ## in *STIC is greater than the number of STL files ## in *STLF.
```

This error occurs when there is a numbering mismatch between *STIC and *STLF

6.1.9 Element Activation Errors

```
Hybrid element activation *DDM! is used without a laser path using *LSRP or *LASR
or a PRM file set by *PBPF. Check and correct the input files.
```

This error is shown when a user includes the hybrid element activation card *DDM! without with a laser file or PRM file.

```
Hybrid element activation *DDM! error:  top substrate coordinate is lower than
substrate based.  Ensure the values are correct an in the correct order.
```

This error occurs when the base of the .stl file floats above the top of the substrate as specified by *DDM!.

```
DDM* elements activated when heat source is active at time= ##.  Check *DDM*
values.
```

Elements cannot be activated when the heat source is active. Adjust the timing used by the *LSRF or *DDM! cards.

```
The homogenized geometry is empty.  Try using a larger alpha radius (maximum cell
radius) or check the input geometry.
```

This error occurs when automatic homogenization has been enabled but the alpha radius has be set so small that no geometry is created. Increase the alpha radius in the *STLH card to correct this problem.

6.1.10 Substrate specification errors

```
Substrate extension *SBXY card cannot contain negative values.
```

This error occurs if negative values are used in the *SBXY card.

```
*SBXY substrate extension and *SBDM substrate dimensions cannot be used
simultaneously.
```

This error occurs with the user attempts to use both *SBXY and *SBDM to extend the substrate geometry beyond the part bounding box.

```
Substrate dimensions set by *SBDM card creates a bounding box outside of the
```

`substrate. Check *SBDM values.`
This error indicates that some of the part lies outside the region described by *SBDM. Double check the dimensions and location of the input geometry and alter the *SBDM card accordingly.

6.1.11 Meshing errors

`*PBPA powder bed layer grouping is used with no *PBPF specified PRM file. Add the *PBPF card with a valid PRM file.`
This error occurs with a part scale simulation is attempted without specifying a .prm file in the *PBPF card.
`Layers per element set by *PBPA is used with no coarsening generations set by *ADAP.`
The maximum number of adaptivity levels must be specified using *ADAP for part level simulations.
`*LSRP automatic laser vector generation is used with layer grouping *PBPA. Remove the *LSRP card for a part scale analysis or *PBPA for a moving source analysis.`
This error occurs when the user includes the *LSRP card during a part scale simulation.
`The number of coarsening generations set by *PBLR cannot be negative. Increase the *PBLR value.`
These errors occur when the value in the *ADAP or *PBLR cards are negative.
`The number of coarsening generations set by *PBLR cannot be less than the adaptivity levels set by *ADAP. Increase the *ADAP value or decrease the *PBLR value.`
The value of *PBLR must be less or equal to the value in *ADAP.
`*ADAP based mesh coarsening can only be used in conjunction with hybrid activation specified by *DDM!`
The *ADAP card requires the use of the *DDM! card.
`*ADPM moving source adaptivity can only be used with thermal analyses, *ANTP 2.`
As of this time *ADPM only works for thermal simulations, so users receive this error if the *ADPM is included in a mechanical input card.
`Zero number of elements in refined mesh. Check if substrate coordinates in hybrid element activation card *DDM! are correct.`
`Number of elements must be positive. Check that the hybrid activation *DDM! settings are correct.`
These errors occur when the final refined mesh matrix is empty which is usually caused by specifying *DDM! values that encapsulate the entire part.
`Model size exceeds max 8-byte signed integer.Increase layers per element set by *PBPA and coarsening generations set by *PBLR to reduce mesh density.`
This error will occur only for very large models which exceed the limits of the 8-byte integer to simulate the process. Reduce the mesh settings with *PBLR and *PBPA and rerun the simulation.

6.1.12 Simulation settings errors

`No analysis type *ANTP defined`
An analysis type, either 2 for thermal or 4 for mechanical, must be specified in every input file using *ANTP.
`Powder bed dwell time set by *PBDL must be positive.`
The powder bed dwell time multiplier specified by *PBDL must be positive.
`*PBDL Powder Bed Dwell time is used without number of layers per element`

`specified by *PBPA.`
Powder bed dwell time *PBDL is applicable only to part level Powder Bed fusion analyses, with proper *PBPA, *ADAP, and *PBLR mesh settings. To add dwell time effects within the PRM generation process, use *TPRM.
`*TRAN transient settings is required with *LSRP auto-generate laser path.`
`*TRAN transient settings is required with layers per element grouped by *PBPA .`

Performing PRM generation simulations with *LSRP or part scale powder bed simulations requires the use of the timing control card *TRAN.

6.1.13 Material property errors

`Material properties for Material ID 1 have not been defined in the input file. Check the formatting of the material properties *MATE and *MATI cards.`
This error indicates that either the *MATE or *MATI cards, or both, is missing from the input file.
`Material props not defined for MATID ##. Check that *MATE and *MATI have been fully defined and that the format of the neutral file is correct.`
When using a neutral file based mesh with *INPU, the following error will be issued if the *MATE and *MATI cards have been used correctly to define the material properties. Ensure that the correct property numbers from the neutral file are used in the material block and that the necessary properties have been included.
`No material properties have been defined. Correctly define the material properties using the *MATE and *MATI cards.`
This error is issued when no material properties have been defined in the input files.
`Melting temperature specified by *MLTT must be higher than the stress relaxation temperature set by *SRLX for material ##`
This error indicates the specified melting temperature (*MLTT) is lower than the specified stress relaxation temperature (*SRLX).
`Cut off temperature specified by *TCUT must be higher than the stress relaxation temperature set by *SRLX for material ##`
`Cut off temperature specified by *TCUT must be higher than melting temperature set by *MLTT for material ##`
These errors indicate that the material property cut off temperature (*TCUT) is lower than either the stress relaxation (*SRLX) or melting (*MLTT) temperature.
`Max number of materials exceeds 100. Increase the value of number of allowable materials *NMAT.`
By default up to 100 different material IDs may be used with *MATI. To increase this limit, use *NMAT.
`Number of temperature dependent material properties exceeds 100. Use *NMTT to increase the maximum number of allowable material property temperatures.`

This error indicates that maximum number of temperature values set by `*NMTT` has been exceeded. The default value is 100 temperatures. Use the `*NMTT` card to increase the number of allowable temperatures.
`Number of yield points exceeds 10. Check that proper number of yield points has been specified in the *PLAS plasticity properties card.`
This indicates the number of allowable plasticity points for the material specified by the `*PLAS`

card has been exceeded. A maximum of 10 pairs of stress-strain points can be used describe the property curve at each temperature.

```
Plastic strain must be entered in order of increasing strain.  Check and fix
*PLAS plasticity properties.
```

This error is displayed when the yield strength and strain properties in the *PLAS card are not ordered in increasing strain for each specified temperature. Check the *PLAS card for typos and correct formatting.

```
Number of yield points must be positive.
```

The number of yield points specified by *PLAS must be greater than zero.

```
Conductivity must be positive.
Density must be positive.
Specific heat must be positive.
Ultimate stress must be positive.
Elastic modulus must be positive.
Poisson's ratio must be between 0 and 0.5.
Yield strength must be positive.
Equivalent plastic strain must not be negative.
Spring constant must be positive.
```

These errors indicate that a material property has be entered in an invalid range. Check the material properties specified in the input files and rerun the simulation.

```
Conductivity properties must be defined using *COND for MATID ##
A density value must be defined using *DENS for MATID ##
Specific heat properties must be defined using *SPEC for MATID ##
Elastic properties must be defined using *ELAS for MATID ##
Thermal expansion values must be defined using *EXPA for MATID ##
```

These material properties must be included.

6.1.14 Boundary condition errors

```
Number of fixture points exceeds 100.  Increase maximum number of material values
set by *NMTT.
Number of spring fixture points set by *FISR exceeds 100.  Increase *NMTT.
Number of symmetry planes exceeds 100.  Increase maximum number of material
values set by *NMTT.
Number of convection points exceeds 100.  Increase *NMTT.
```

These errors occur when the number of boundary conditions specified in an input file exceeds the default limit of 100 values. This value may be changed using the *NMTT card.

```
Rectangular fixture card *FIXR values must be ascending, ##.
Spring fixture *FISR card values must be ascending, number ##.
Rectangular prism fixture card *FIxZ values must be ascending, number ##.
```

These errors occur when the mechanical boundary conditions are not sorted in ascending order.

```
*FIXR card rectangular fixture outside substrate, number ##.
*FIXC card circular fixture outside substrate, number ##.
```

These errors occur when one or more mechanical boundary conditions are specified outside of the

substrate bounding box.
`Less than 6 dispalcements were fixed. Adjust the dimesnions of the boundary condition cards or remove custo BC cards to allow the solver to automatically fix 6 displacements by default.`
This error is shown when the mechanical constraints are not sufficient to prevent transitional or rotational motion.
`Non-positive load case number in transient boundary condition set by *RAMP. Boundary condition ID ##. Check the load case number.`

This error occurs when a negative load case number is used in the `*RAMP` card.
`Ramp no ## not implemented`
This error will be displayed if the user specifies a RAMP ID number that is not supported by the solver.
`Time dependent boundary condition *RAMP no. ## not implemented.`
This error indicates the chosen *RAMP condition is not a valid boundary condition.
`Symmetry plane ##... is not aligned with substrate bounds. Check *SYMM, *SBDM, and *SBXY, or us *SYM2 to enable misaligned symmetry planes.`
Symmetry boundary conditions using *SYMM must be aligned with the substrate.

6.1.15 Output errors

`Output file size *OFNS value is less than or equal to zero. Increase *OFNS value in the input files.`
This error indicates a zero or negative value was used in the *OFNS card.
`Output file size *OFNS value is greater than 10. Decrease *OFNS value in the input files.`
The maximum value allowed by the *OFNS card is 10 to ensure memory constraints are not exceeded.
`Cannot use negative values in auxspar scaling factor *AXSP. Increase the *AXSP scaling factor.`
If the user manually specifies the auxspar scaling factor using *AXSP, it cannot be less than 0.
`Restart file cannot be the same as input file. Rename the current input file and rerun.`

This error occurs when the restart file name specified by *REST is identical the the input file name in which the card is written. Rename the input or restart file name.
`Restart files can only be used with the same version. This file was generated by v##.##.##`

Restart files can only be used using the same version of the solver as the restart file was originally generated.

6.1.16 Convergence errors

`Residual is too big. Check solution settings specified by *SOLU. Convergence may be caused by extremely non-linear material properties or poor processing conditions.`
This error warns the user the residual is above the tolerance specified in *SOLU. The simulation

will reduce the time step and attempt to continue the simulation. If this process occurs more times than maximum number of cutbacks allowed (also specified by the *SOLU) card, the simulation will abort.

Best Practices

1. Thermal Simulations - To avoid convergence errors: i. Increase the width of the `*LATE` Liquidus and Solidus temperatures. ii. Turn on relaxation, *SRLX with values 1, 0.4. If convergence continues to be problematic, increase this to 2,0.4

2. Mechanical Simulations - To avoid convergence errors i. Turn on relaxation, *SRLX with values 1, 0.2. If convergence continues to be problematic, increase this to 2,0.2

`Too many increments. Aborting Execution. Increase maximum number of allowable increments in *TRANS.`

This error occurs if the number of increments specified by the *TRAN card is exceeded. Increase the maximum number of increments and rerun. Note: the number of output files in *OFNS may also need to be increased.

`Too many cutbacks. Aborting Execution. Check material properties and processing parameters. Consider adding or increasing Newton-Raphson relaxation set by *RELA.`

This error occurs if the number of cutbacks specified by the *TRAN card is exceeded. Cutbacks are caused by poor convergence. These can be improved by introducing or increasing solver relaxation via *RELA. The number of cutbacks may also be increased.

Plasticity algorithm fails to converge. Check material properties and processing parameters. Consider adding or increasing Newton-Raphson relaxation set by *RELA.

`The time step required to solve is less than the minimum allowable time step specified in the *TRANS card. Rerun with numerical relaxation or reduce the minimum allowable time step.`

This error occurs when the solver takes a series of cutbacks, reducing the time step to step forward to the next increment. If the time step the solver attempts to use is less than the minimum time step allowed in the *TRAN card, the solver will abort.

`Error: Solver failed to converge using Large Deformation Formulation. Buckling may have occurred.`

This error is shown in the case when a part scale simulation fails to converge and the Large Deformation card, *NLTL, has been enabled. This error indicates that the geometry and processing conditions are liable to buckle.

6.1.17 Patran mesh errors

`Both 2D and 3D elements input file`

This error indicates that there are still 2D elements in the voxel mesh generated in Patran. Delete these elements and resave the file to proceed.

6.1.18 Laser file errors

```
*LSRP auto-generated laser path and *LASR laser path card are used
simultaneously.
*LSRP auto-generated laser path and *LASF laser path file are used
simultaneously.
```

These errors indicate the user attempted to automatically generate a laser vector file with *LSRP

while also trying to manually input a .lsr file via *LSRF. Laser vectors can only be defined using one method per simulation.

`Cannot open file example.lsr. Check that the .lsr file is in the correct directory and has been specified correctly in the input files.`

This error indicates the specified .lsr file is either misspelled or in the wrong directory.

`Cannot read file example??.lsr. Laser line: ## is incorrectly formatted.`

This error lets the user know the specified laser line in the .lsr file is not formatted correctly.

`No layers in laser path`

This error occurs when the laser file specified by *LSRF has zero entries.

`*LASR laser vector card lines are not in time ascending order. Line: laser-line##`

This error occurs when the laser paths specified in the laser file *LSRFare are not sorted in chronologically increasing order.

`Laser line has zero length. Laser line: ##. Check and fix .lsr file.`

This error occurs when a laser path specified in the laser file *LSRF has identical beginning and ending coordinates.

`Zero or negative travel speed for weld line ##. Increase travel speed in the .lsr file.`

Travel speeds in the .lsr file specified by *LSRF or created by *LSRP must be positive. Correct the .lsr file and rerun the simulation.

`Zero or negative radius for laser line ##. Increase laser radius in the .lsr file.`

This error indicates that the heat source radius is equal or less than 0 mm. Positive heat source radii must be used. Correct the .lsr file and rerun.

```
Path error: travel speed must be positive. Check and fix .lsr file.
Path error: layer thickness must be positive. Check and fix .lsr file.
Path error: hatch spacing must be positive. Check and fix .lsr file.
Path error: dwell time must be positive. Check and fix .lsr file.
Path error: number of layers must be positive. Check and fix .lsr file.
Path error: x width must be positive. Check and fix .lsr file.
Path error: y width must be positive. Check and fix .lsr file.
```

These errors are issued when the .lsr has sets any of the critical values to a zero or negative value. Fix the *LSRP values.

`*STLF is required with *LSR3`

For 6 Axis DED simulation a substrate STL file must be assigned via *STLF

6.1.19 Heat treatment errors

`Stress relaxation time history requires more than one point.`

`Heat treatment schedule card *STRF requires time values to be non-negative. Input a valid heat treatment schedule.`

`Heat treatment schedule card *STRF requires the heat treatment history to be specified. Input a valid heat treatment schedule.`

`Heat treatment schedule set by *STRF card requires the time values to be entered in ascending order.`

These errors occur when the time history specified by *STRF is invalid. Heat treatment schedules must have at least 2 time-temperature sets which increase in time ascending order. Correct the

heat treatment schedule and resubmit the simulation.

```
Stress relaxation *STRF can only be used when part-level plasticity *PPLA is on.
*STRF card will be ignored.
Stress relaxation *STRR can only be used when part-level plasticity *PPLA is on.
*STRR card will be ignored.
```

These errors indicate that the user attempted to enter in a heat treatment relaxation temperature or temperature history without enabling part-level plasticity, using *PPLA. Add the *PPLA card to the mechanical input file and rerun the simulation.

```
Stress relaxation temperature is specified without the heat-treatment
time-history.
Stress relaxation temperature is not specified for heat treatment.
```

These errors occur when the heat treatment history *STRF has been used without a heat treatment stress relaxation temperature *STRR or *STRR without *STRF. Make sure the heat treatment history has been added to both the thermal and mechanical input files and that *STRR has been included for each of the material properties specified.

6.1.20 End of Simulation Errors

```
Recoater interference.
```

This error only occurs if Recoater Interference stoppage is enabled in the *RCTR card, which indicates recoater interference has been predicted and the simulation has been truncated.

```
Wall time = ## exceeds maximum wall time = ##.  Check maximum wall time set by
*TWAL.
```

This error occurs when the CPU wall time has exceeded the maximum time allowed by *TWAL. If the simulation ended prematurely change the value of *TWAL or remove the card from the input files and rerun the simulation.

```
The maximum number of iterations ## has been exceeded and the simulation has been
terminated.  If this has occurred unexpectedly check for convergence errors and
adjust the maximum number of iterations allowed.
```

This error occurs when the number of iterations has exceeded the maximum number allowed by *TRAN. If the simulation ended prematurely change the value of the maximum iterations allowed set in the *TRANS card.

6.2 Critical Warnings

6.2.1 System Critical Warnings

```
There may not be enough available physical memory to complete the analysis.
Ensure other programs are not running.  Reduce the mesh density if necessary or
run on a workstation with more memory.
```

This critical warning occurs during the mesh generation process, which includes an estimate of maximum memory usage for the simulation, indicating the simulation may exceed the system memory and will not complete successfully.

```
Running on an encrypted drive will increase simulation time.  Consider solving on
a non-encrypted drive to improve performance.
```

```
Running on a network drive will increase simulation time.  Consider solving on an
internal drive to improve performance.
Running on a USB drive will increase simulation time.  Consider solving on an
internal drive to improve performance.
```

These critical warnings are shown whenever the simulation is run on an encrypted, network, or USB drive. These drive types can cause small to significant increases in simulation time. It is recommended that users always simulate on a non-encrypted, internal drive, with the highest availed transfer and write speeds to ensure system performance does not artificially slow Netfabb Simulation .

6.2.2 PRM File Critical Warnings

```
PRM file example1??.prm was generated by an older version of Autodesk Netfabb
Local Simulation.  Consider regenerating the PRM file with the latest version of
Autodesk Netfabb Local Simulation.
```

This critical warning occurs when the PRM file version does not match the current version of the PRM generation tool. It is strongly recommended the PRM file be regenerated to ensure the PRM file is up to date and includes all available features.

```
PRM file example??.prm does not contain conductivity properties.  Ensure the input
file has conduction properties or regenerate the PRM file with the correct properties.
PRM file example??.prm does not contain density properties.  Ensure the input file
has density properties or regenerate the PRM file with the correct properties.
PRM file example??.prm does not contain specific heat properties.  Ensure the input
file has specific heat properties or regenerate the PRM file with the correct properties.
PRM file example??.prm does not contain elastic properties.  Ensure the input file
has elastic properties or regenerate the PRM file with the correct properties.
PRM file example??.prm does not contain thermal expansion properties.  Ensure the input
file has thermal expansion properties or regenerate the PRM file with the correct properties.
PRM file example??.prm does not contain plasticity properties.  Ensure the input file
has plasticity properties or regenerate the PRM file with the correct properties.
```

These critical warnings occur when a PRM file is used which is missing key material properties.

```
PRM files that contain material properties cannot be combined with those without
material properties.  Rerun using compatible PRM files.
```

This critical warning indicates that one or more PRM files with material properties are used with one or more PRM files without material properties.

```
Material properties in prm file example1??.prm do not match example2??.prm
```

This critical warning indicates that multiple PRM files were used with mismatched material properties. Using multiple materials in a powder bed machine is not yet feasible. The properties from the first PRM file will be used for the simulation.

6.2.3 STL file and Meshing Critical Warnings

```
STL is too small.  Check units.  This warning is triggered when any bounding box
dimension of the STL is smaller than the base element size.  Possible causes are:
1 - The STL file is too small.  Make sure that the units for the STL are correct.
```

```
2 - A course mesh size is used.  Try refining the mesh size.
```

This critical warning indicates that the STL file may be in the wrong units or poor mesh settings were used.

```
STL file example??.stl could not be repaired.
```

By default Netfabb Simulation attempts to repair non-manifold STL files using the default Netfabb repair script. This critical warning notes whenever this repair process fails.

```
More than 20 percent of the mesh was removed.  Increase the mesh density, add
support structures, and check the component(s) are oriented so as to maintain
component-build plate contact.
```

At the beginning of each grouped powder bed layer, elements generated for that region during the mesh preview are activated. If the elements are not connected to the baseplate or to previously meshed component to prevent translation and rotation, the elements are considered erroneous as parts cannot be constructed completely detached from the material below, and these elements are removed from the simulation. If more than 20% of the mesh preview elements are removed from the simulation, this Critical Warning is issued, indicating that the STL has been oriented or supported poorly.

```
## percent of the mesh is unsupported.  Increase the mesh density, add support
structures, and check the component(s) are oriented so as to maintain
component-build plate contact.
```

This error occurs when more than 1% of the original geometry volume is not meshed.

6.2.4 Convergence Critical Warnings

```
Plasticity algorithm did not converge after ## iterations.  Check material
properties and processing conditions.  Increase the number of relaxation
iterations.
```

This critical warning is shown if the plasticity step fails to converge. This will generate a cutback.

```
Residual is increasing.  Reducing time step.
```

This critical warning indicates the residual for the current increment increased, indicating a lack of convergence. This will generate a cutback.

```
Residual is too large.  Reducing time step
```

This critical warning indicates the residual for the current increment is higher than the specified limit, indicating a lack of convergence. This will generate a cutback.

```
The number of equilibrium iterations has exceeded the maximum allowed number of
equilibrium iterations.  Reducing time step.
```

This critical warning occurs during heat treatment analysis when the solver fails to converge. This will generate a cutback.

6.2.5 Simulation settings Critical Warnings

```
Part scale plasticity is enabled without plasticity properties assigned for build
plate material ##.  Stresses will be qualitative and post EDM displacements will
be exaggerated.
```

When *PPLA part scale plasticity is enabled this critical warning is generated if the build plate properties are specified using *MATE and *MATI which do not contain the *PLAS card. Lacking these properties, the build plate will not be able to yield during the post-process plasticity step, which will lead to excessive stresses and distortions.

```
CLI type support structures have been deprecated due to poor meshing behavior.
Consider replacing the CLI support file with a STL support file.
```

CLI support structures are not recommended for use with Netfabb Simulation .

```
Powder bed side convection set by *PBCP is used without powder properties being
enabled via *+PDR. *PBCP convection values will be ignored.
```

Powder bed side convection *PBCP requires powder elements to be enabled via *+PDR.

```
Substrate bottom convection set by *PCSB is used with a constantly heated
substrate, controlled by *PBSH. *PCSB convection values will be ignored.
```

Powder bed bottom convection *PCSB will be overridden by using a heated substrate, set by *PBSH.

```
Substrate side convection set by *PCSS is used with an insulated substrate set by
*PBIS. *PCSS convection values will be ignored.
```

Using an insulated substrate by *PBIS will not apply any substrate side convection set by *PCSS

```
Support structure failure detect at ## percent of support-component interface
elements.
```

This warning will occur at the end of a part-scale simulation which exhibits support structure failure indicating how much of the contact support volume has failed as a percentage.

```
*FIxZ line ## did not capture nodes.  Increase the bounding coordinates of the
box.
```

This warning informs the user the *FIxZ boundary condition settings used fails to constrain any nodes.

6.2.6 Heat source Critical Warnings

```
Laser line has zero length
## ignored
```

Laser lines imported by *LSRF or generated using *LSRP which have zero, or close to zero laser length, will not be processed by the solver.

```
More than one heat source detected operating simultaneously.  If this is not a
multi-laser system check the .lsr file for errors.
```

This critical warning occurs when 2 or more simultaneous heat sources are active. If this was not the design intent, fix the .lsr file.

6.3 Warnings

Each warning is proceed by a flag and a counter which looks like:

```
**********************
number-warnings## Warning
**********************
```

Which informs the user of how many total warnings have been recorded for the present simulation.

6.3.1 Run time warnings

```
-f option is ignored without *PBPA
```

Fast costing mode using the -f option is restricted to part-scale powder bed fusion models. Other modeling types will use the standard -c costing mode instead.

6.3.2 Material property warnings

```
Card-name??  is used outside material block
```

This warning lets the user know the input file being run contains the material property card Card-name?? outside the material property block and will not include these material properties.

```
Material id=0 for element:  element-number##
Material id set to 1
```

When a Patran generated mesh is used, this warning indicates some elements were not properly assigned a material id. Netfabb Simulation changes these to property id 1 by default.

```
*PBS2 and *PBSN cannot both be used.
*PBS2 is ignored.
```

If *PBSN and *PBS2 are both in the input file, the former is followed.

```
*COND, *DENS, *SPEC, *ELAS, and *EXPA
are ignored for *MATID ##
The properties in prm file
example??.prm
will be used instead.
```

This warning indicates material properties were written for build component material in a part scale input file. These will not be read, as the material properties for components are now written to and read from the PRM files.

```
*NMAT must be positive
'Resetting to 100
```

This warning occurs when a negative value is used in the **NMAT card.

```
*NMAT is ignored if it is not the
first card in the input file
```

This warning occurs when *NMAT has been included but not as the 1st card in the input file.

```
*NMTT must be positive
'Resetting to 100
```

This warning occurs when a negative value is used in the **NMTT card.

```
*NMTT is ignored if it is not the
first card in the input file
```

This warning occurs when *NMTT has been included but not as the 1st card in the input file.
`Number of materials exceeds maximum allowed. Increase number of allowed materials with *NMAT and rerun.`
This error occurs if the user specifies more than 100 materials in an input file. Increase the maximum allowable number of material using the *NMAT card.

6.3.3 Boundary condition warnings

`Laser line laser-path-line## has ended before start time`
`Laser line deactivated`
This warning informs the user the start time in the *TRAN card is after one or more lines in the laser path (specified by *LSRF).
`Laser line length is less than 10% of laser radius`
Laser lines of negligible length, much smaller than the laser radius itself, are non-physical and may lead to erroneous results. A few of these are commonly generated using certain *LSRP settings.
`Convection removed from internal face of element element-number##`
Patran generated meshes can often have convection assigned to interior faces. This warning informs the user that Netfabb Simulation has automatically removed convection boundary conditions from one or more interior faces.

6.3.4 Mesh warnings

`## percent of the mesh is unsupported. Increase the mesh density, add support structures, and check the component(s) are oriented so as to maintain component-build plate contact.`
This warning occurs if any of the mesh is going to be removed for being loose, completely unsupported elements, or partially connected rotating elements. Elements which will be removed are shown in the mesh preview Structure Type results with the numeric value -1.00 or the title "Unsupported Elements." If the stated percentage is significant, use the log information during meshing and the Structure type results to improve the orientation or support structures to reduce or eliminate unsupported elements.
`disconnected-element-number## Loose or rotating elements removed`
This warning indicates the numbered element is not connected to the existing part or substrate, which would result in a non physical floating mass, and has been removed from the mesh. Rotating the build volume or adding support structures may eliminate these warnings.
`Negative or zero *NELR. Reset to 1`
This warning indicates the user has specified less than 1 element per laser radius. Netfabb Simulation then automatically changes *NELR to the default value of 1.
`Number of refinement levels must be positive.`
*ADP1, which specifies the number of layers below the current layer to keep fine, must be positive.
`Node node-number## not assigned no dof will be added`
This warning informs the user the node number called out will not be added to the solution matrix.
`Element type is not available, type= element-type## elemnum=element-number##`
Patran generated meshes allow for many element types, but currently only hex8 elements are compatible with Netfabb Simulation . This warning informs the user the Patran mesh has one or more non-compatible element types.
`TOO LARGE NON-CONTINOUS NODE NUMBER, PLEASE RENUMER`

Patran generated meshes may have gaps in their node numbering schemes. If a gap is too large it may slow down simulations and it is suggested the user renumber the mesh nodes.
`TOO LARGE NON-CONTINOUS ELEM NUMBER, PLEASE RENUMER`
Patran generated meshes may have gaps in their element numbering schemes. If a gap is too large it may slow down simulations and it is suggested the user renumber the mesh elements.
`*STOL is ignored for a single STL`

When just one STL file is specified and *STOL is included in the input file, it will be ignored to prevent causing artificial meshing errors.
`Reseting *ADAP to 1 for moving source model`
This warning occurs when *ADAP is set to 0 during moving source models, which is overridden and set to 1 to avoid poor mesh performance.

6.3.5 Build failure warnings

```
 Recoater clearance = clearance-percentage## %
Top z deformed coord:  z-max-coordinate## Recoater coord:
```
This warning indicates build failure could occur due to recoater interference, as the recoater clearance is less than either the default value of 20% or the user specified tolerance controlled by the *RCTR card.
```
Support structure failure
------------------------------
```
This warning indicates a support structure element has failed according to the failure stress specified by the *UTSR material property card.
`Support structure interference detected at ## % of support-component interface elements.`
The warning, shown at the end of a mechanical simulation log where support failure occurs, indicates that support failure occurred and how much of the support-component interface has failed.

6.3.6 Heat treatment model warnings

```
*STRF card requires the time values to be positive.
*STRF card will be ignored

*STRF card requires the time values to be
entered in ascending order
*STRF card will be ignored

*STRF card requires the heat treatment history
to be specified
*STRF card will be ignored
```

These warnings occur when the heat treatment modeling history is improperly specified. If this occurs the simulation will run but will not perform the heat treatment modeling.
```
Stress relaxation *STRF can only be used when
part-level plasticity *PPLA is ON
*STRF card will be ignored
```

This warning occurs when a user attempts to model heat treatment without using part-scale plasticity. If this occurs the simulation will run but will not perform the heat treatment modeling.

6.3.7 Output warnings

`*BINA is only used for thermal models`
This error indicates the *BINA card was erroneously written into a mechanical input file.
`Time is too close to previous time for postprocessing. Postprocessing will show artificially increased times from this increment onwards`
This warning occurs when a time step is identical to the previous time step, up to 5 significant figures. To improve post processing, a 1 is added to the last significant figure in the simulation result time stamps. However the timing used for the simulation itself is not altered.

References

[1] P Michaleris. Modeling Metal Deposition in Heat Transfer Analyses of Additive Manufacturing Processes. Finite Elem in Analysis and Design, 86:51–60, 2014.

[2] JC Heigel, P Michaleris, and EW Reutzel. Thermo-mechanical model development and validation of directed energy deposition additive manufacturing of ti–6al–4v. *Additive Manufacturing*, 2014.

[3] John Goldak, Aditya Chakravarti, and Malcolm Bibby. A new finite element model for welding heat sources. *Metallurgical transactions B*, 15(2):299–305, 1984.

Index

*+PDR, 48
*3PLP, 77
*ACNV, 95
*ADAP, 40
*ADDP, 27
*ADP1, 42
*ADP2, 43
*ADP3, 43
*ADPM, 41
*AMBI, 71
*ANNL, 68
*ANTP, 31
*AUBC, 70
*AUTM, 85
*AVTK, 26
*AXSP, 31
*BINA, 24
*BMUL, 57
*BTOL, 57
*CAS1, 27
*CCNV, 72
*CLIF, 90
*CMSH, 31
*COND, 64
*CONV, 72
*COOL, 85
*DASH, 64
*DDM!, 37
*DDM1, 37
*DDM6, 39
*DDMC, 38
*DDMM, 38
*DDMP, 38
*DDMZ, 39
*DENS, 65
*DEPE, 24
*DSHP, 64
*ELAS, 65
*EMIS, 65
*EMUL, 57
*END, 31
*ENSA, 26
*ENSF, 27
*ENSI, 26
*ETOL, 57
*EVAP, 68
*EXPA, 65
*FINT, 71
*FISR, 84
*FIXC, 82
*FIXR, 82
*FIxC, 82
*FIxR, 83
*FIxZ, 84
*FSBT, 52
*FSRD, 84
*FSUB, 51
*GOLD, 76
*GTAB, 63
*GTOL, 79
*HMAT, 63
*HPRM, 63
*HTAT, 95
*HTOR, 95
*INIT, 71
*INPU, 23
*INTA, 28
*IOBN, 23
*KTMP, 33
*LATE, 66
*LENS, 76
*LFUS, 27
*LINQ, 77
*LSR2, 81
*LSR3, 39
*LSRF, 81
*LSRP, 79
*MATE, 63
*MATI, 63
*MAXM, 33
*MLTT, 67
*MNAM, 69
*MNHV, 56
*MNLR, 48
*MPLI, 30
*NAPL, 63
*NBBX, 44
*NBLT, 81
*NELN, 90
*NELR, 87
*NLTL, 31
*NMAT, 69
*NMTT, 69
*NOFC, 32

*NSTP, 56
*NTFE, 53
*NTFT, 53
*ODTX, 26
*OFC1, 32
*OFC2, 32
*OFC3, 32
*OFLX, 26
*OFNS, 25
*OOPP, 25
*ORCS, 60
*ORES, 25
*OSD3, 29
*OSIG, 29
*OSP3, 29
*OSVM, 29
*OVTK, 26
*OWFC, 25
*PARL, 89
*PBCH, 60
*PBCP, 74
*PBCT, 73
*PBDL, 60
*PBIS, 48
*PBLR, 61
*PBPA, 46
*PBPF, 48
*PBS2, 49
*PBSH, 49
*PBSN, 49
*PBSS, 49
*PBSX, 50
*PBSY, 51
*PCSB, 74
*PCSS, 74
*PLAS, 66
*POOL, 26
*PPLA, 29
*PTYP, 70
*RAMP, 70
*RBMO, 85
*RCTR, 58
*RELA, 33
*RELM, 33
*REST, 24
*RSTL, 58
*SBCC, 85
*SBDM, 89
*SBXY, 89
*SOLU, 34
*SPA8, 34
*SPEC, 67
*SPSH, 55
*SRLX, 68
*SSSS, 51
*STAT, 28
*STIC, 54
*STL1, 56
*STL2, 56
*STL3, 55
*STLC, 54
*STLF, 53
*STLH, 54
*STLL, 55
*STLM, 53
*STLS, 57
*STOL, 56
*STR1, 95
*STR2, 96
*STRF, 94
*STRR, 94
*STYP, 24
*SUB2, 43
*SYM2, 44
*SYMM, 43
*TAMB, 71
*TAUT, 35
*TCNV, 73
*TCUT, 69
*TITL, 23
*TPRE, 28
*TPRM, 35
*TRAN, 34
*TWAL, 36
*UNIO, 57
*UTSR, 92
*VCLI, 92
*WRTU, 27
*ZDWL, 35

Execution and File Notation, 17

Getting Started, 7

Keyword Listing, 23

prmgen, 19