

Netfabb Local Simulation

Keyword User Manual

Autodesk Inc.^{[1](#)}

Version 18.3

©February 28, 2018

¹Email: projectpansupport@autodesk.com

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

Contents

1	Overview of the Manual and Netfabb Simulation	7
1.1	Outline	7
1.2	Getting Started	7
1.3	Recommended minimum system requirements	8
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	9
1.6	Netfabb Simulation Capabilities	9
1.6.1	Powder-Bed Part-Level	10
1.6.2	Powder-Bed Fine-Scale	15
1.6.3	Directed Energy Deposition	17
1.7	Pre-processing	19
1.8	Post-processing	19
2	Execution and File Notation	21
2.1	Execution	21
2.1.1	From the file browser (Windows only)	21
2.1.2	From the Windows Command Line or Linux terminal	21

2.1.3	PRM file generation execution	24
2.1.4	Interrogating prm files to determine prm type	26
2.2	File extensions	26
2.2.1	Input Files	26
2.2.2	Output Files	27
3	Keyword Listing	29
3.1	Variable Type Notation	29
3.2	System checks	29
3.3	Input/Output	30
3.3.1	Part Level Stress Output	36
3.4	Analysis control	38
3.5	Element Activation Strategies	45
3.6	Part-Level Powder-Bed Modeling	53
3.6.1	*NTFT	62
3.7	Material Property Definition	73
3.8	Boundary Conditions	80
3.8.1	Temperature Boundary Conditions	82
3.8.2	Convection Boundary Conditions	83
3.8.3	Heat Source Boundary Conditions	87
3.8.4	Mechanical Constraints	94
3.9	Additional Options	98
3.10	Support Structure Modeling	103

4	Post processing tools	109
5	Units	115
6	Common errors and warnings	117
6.1	Errors	117
6.1.1	Autodesk Netfabb Local Simulation LT Errors	117
6.1.2	Runtime errors	118
6.1.3	Licensing errors	118
6.1.4	Netfabb Simulation LT errors	119
6.1.5	Basic input file errors	120
6.1.6	Thermal-mechanical input file mismatch errors	122
6.1.7	PRM generation errors	123
6.1.8	Part scale geometry errors	125
6.1.9	Element Activation Errors	127
6.1.10	Substrate specification errors	127
6.1.11	Meshing errors	128
6.1.12	Simulation settings errors	129
6.1.13	Material property errors	130
6.1.14	Boundary condition errors	132
6.1.15	Output errors	132
6.1.16	Convergence errors	133
6.1.17	Patran mesh errors	134
6.1.18	Laser file errors	134

6.1.19	Heat treatment errors	136
6.1.20	End of Simulation Errors	137
6.2	Critical Warnings	137
6.2.1	System Critical Warnings	137
6.2.2	PRM File Critical Warnings	138
6.2.3	STL file and Meshing Critical Warnings	139
6.2.4	Convergence Critical Warnings	139
6.2.5	Simulation settings Critical Warnings	140
6.2.6	Heat source Critical Warnings	141
6.3	Warnings	141
6.3.1	Run time warnings	141
6.3.2	Material property warnings	142
6.3.3	Boundary condition warnings	143
6.3.4	Mesh warnings	143
6.3.5	Build failure warnings	144
6.3.6	Heat treatment model warnings	145
6.3.7	Output warnings	145

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
#
OMP_NUM_THREADS:4
KMP_STACKSIZE:500000000
```

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.

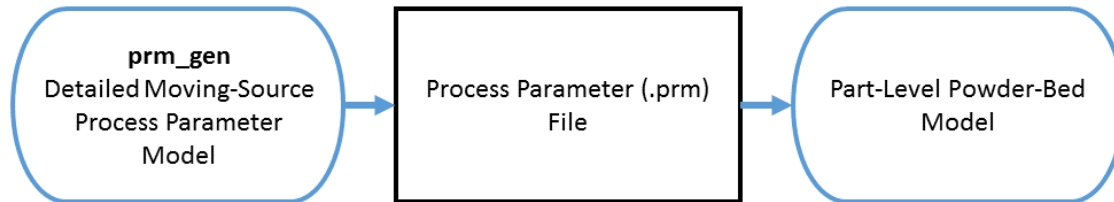


Figure 1.1: Netfabb Simulation Process Parameter (.prm) file generation.

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

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

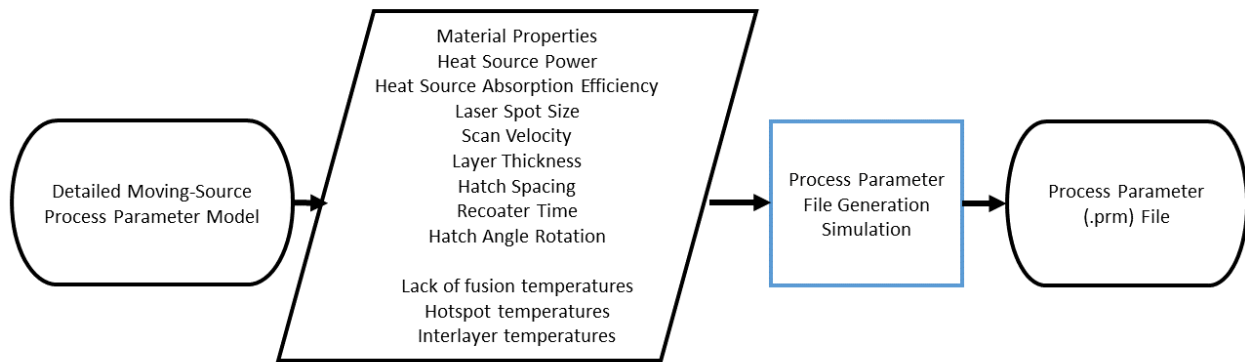


Figure 1.2: Netfabb Simulation Detailed Process Parameter Model.

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

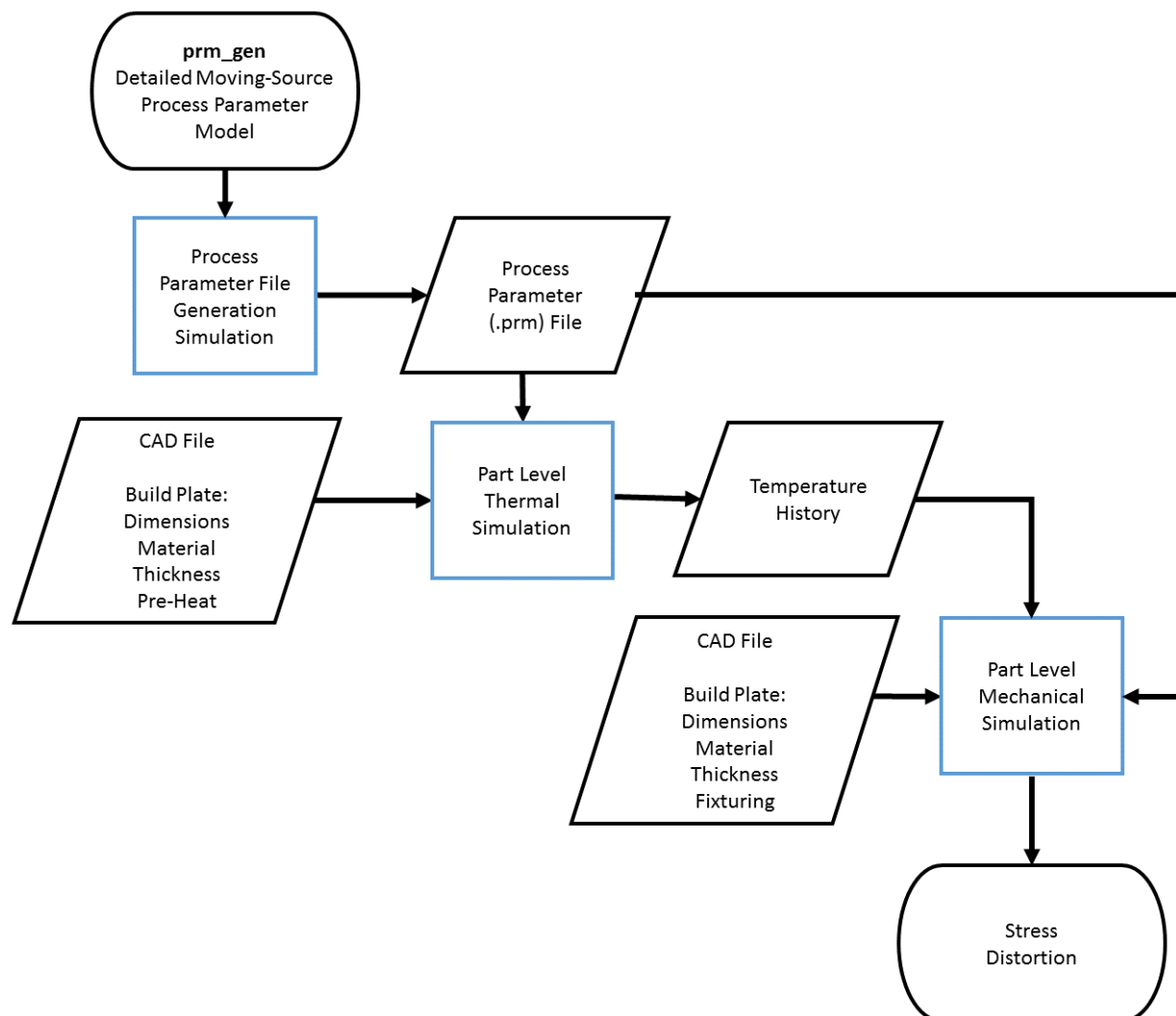


Figure 1.3: Netfabb Simulation Powder-Bed Part-Level Model.

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](#).

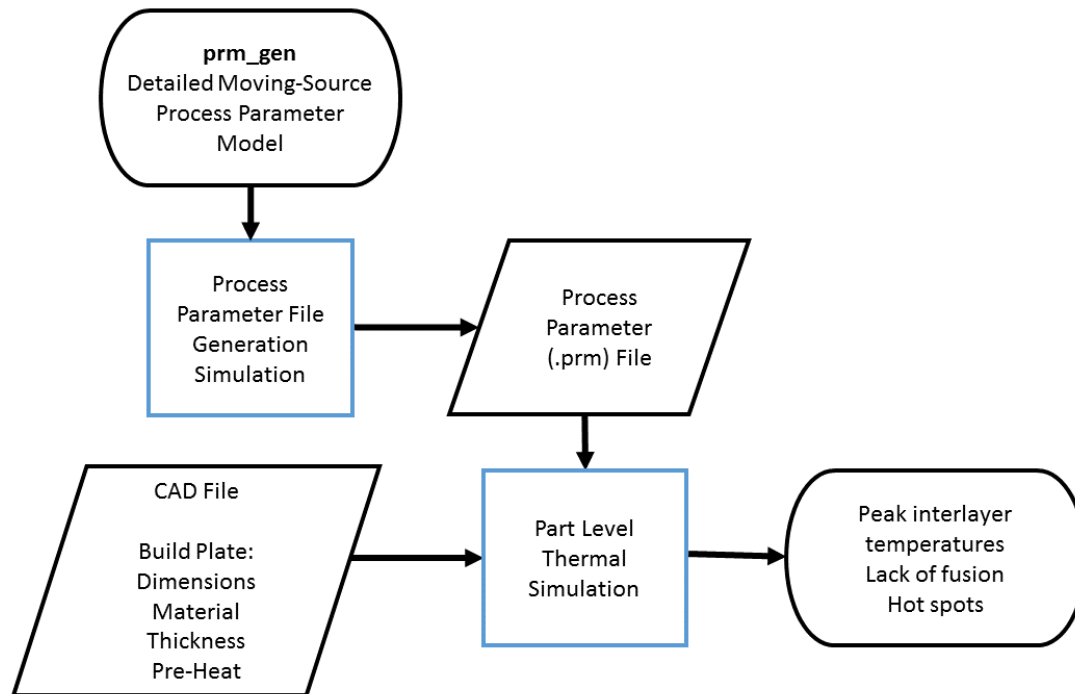


Figure 1.4: Netfabb Simulation Powder-Bed Part-Level Model.

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.

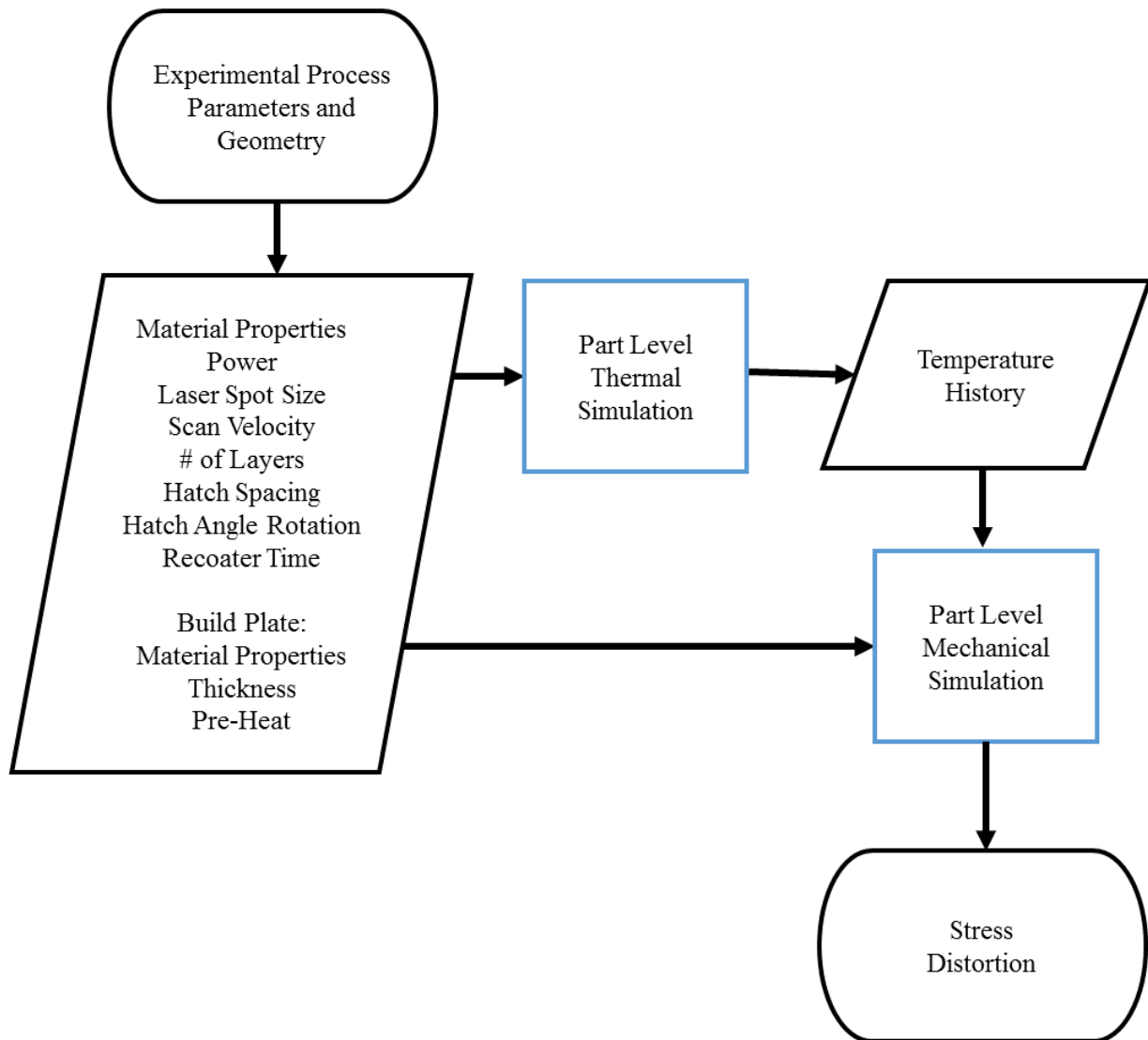


Figure 1.5: Netfabb Simulation Powder-Bed Fine-Scale Model.

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.

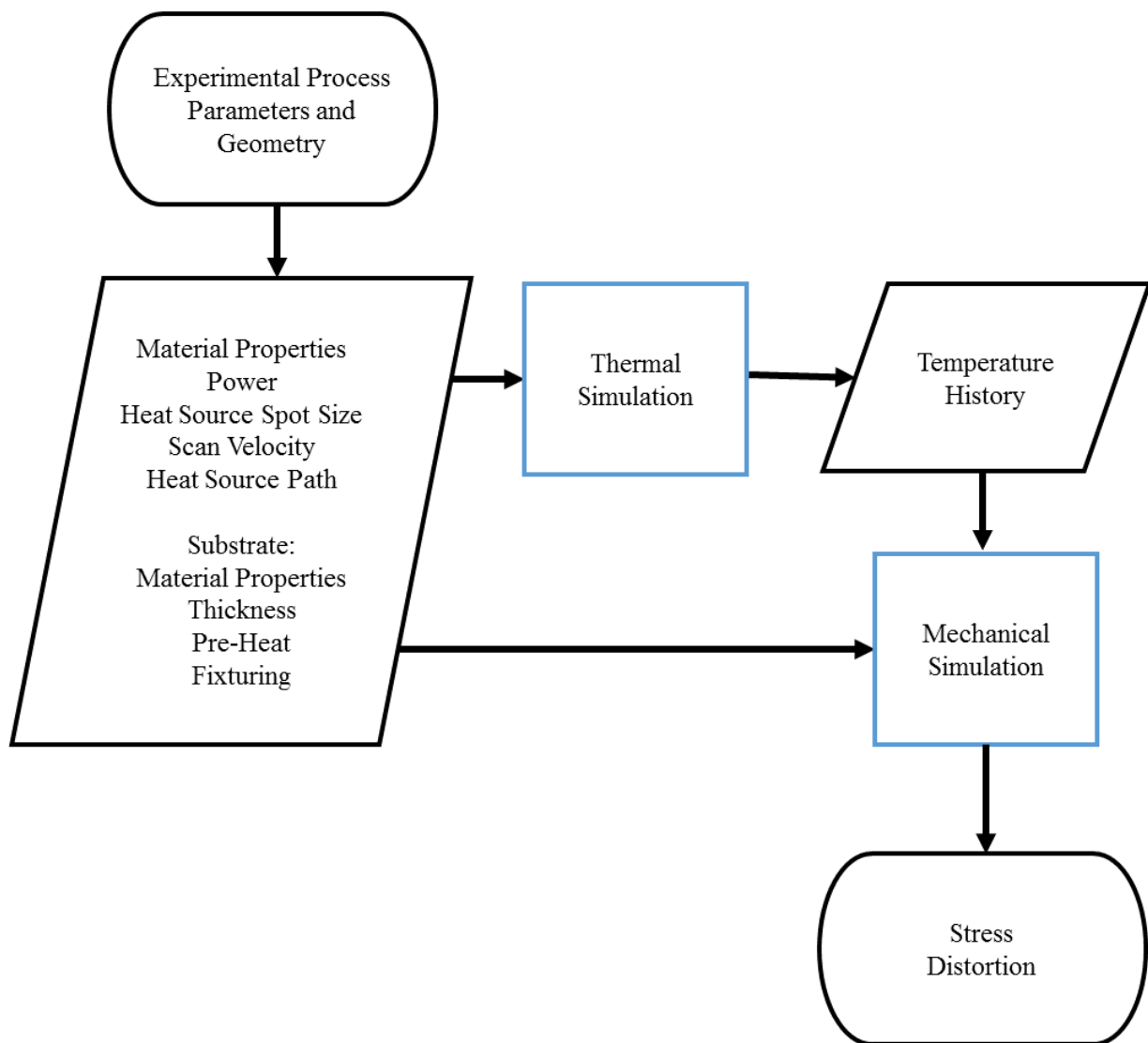


Figure 1.6: Netfabb Simulation DED Model.

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 filename.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 severe 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.

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 not 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
```

2.2 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.

***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 flux switch**

*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.

***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.

***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 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 Enight 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.

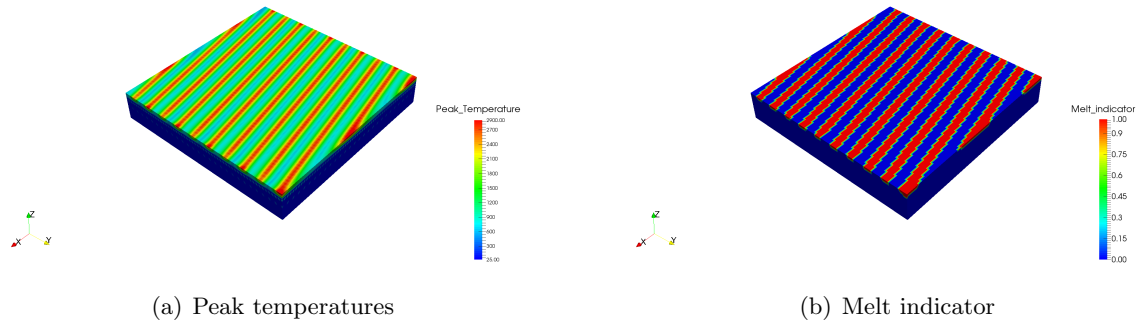


Figure 3.1: *LFUS results

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.

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.

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.

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 $1.10 \cdot r2$. 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 $r1 = 1.1$, $r2 = 1.1$.

***CMESH: Check MeSH**

*CMESH

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.

***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 matrices 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.

***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

il: 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

***SOLU: SOLUtion Parameters**

*SOLU
il, r1, r2

il: 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 il 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 $\text{radius}/\text{velocity} \times r1$, 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.

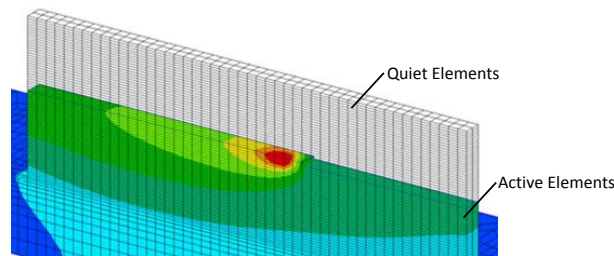


Figure 3.2: Illustration of element activation

***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.

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.

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 `il` allowed is 6. Levels of refinement are demonstrated below:

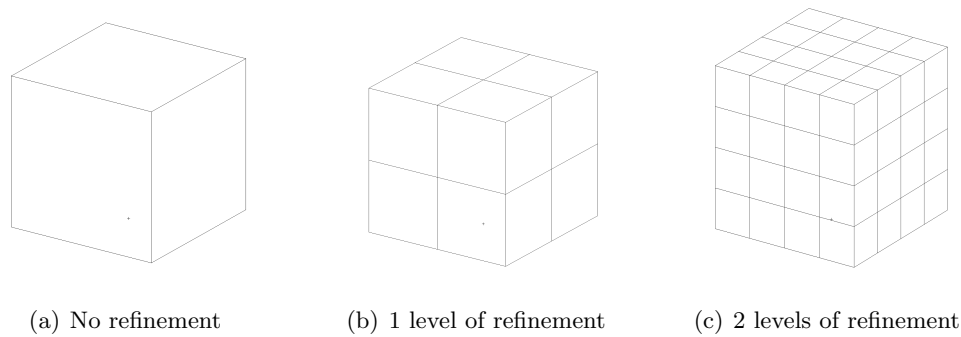


Figure 3.3: Hex8 element subject to various levels of refinement.

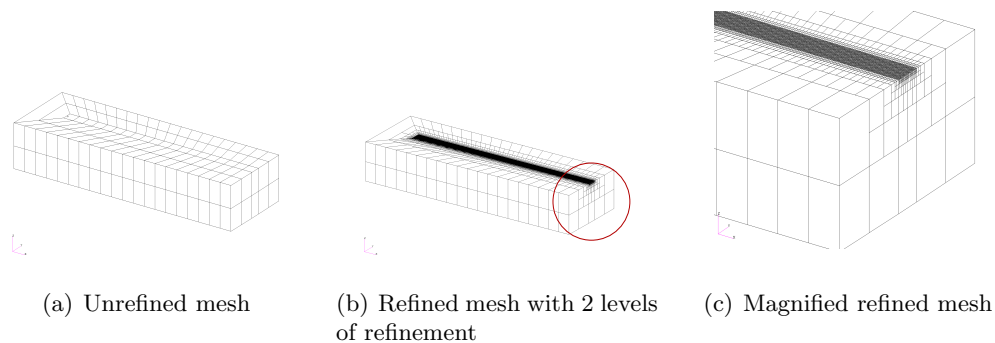


Figure 3.4: Mesh subject to refinement.

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 aberrations 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.

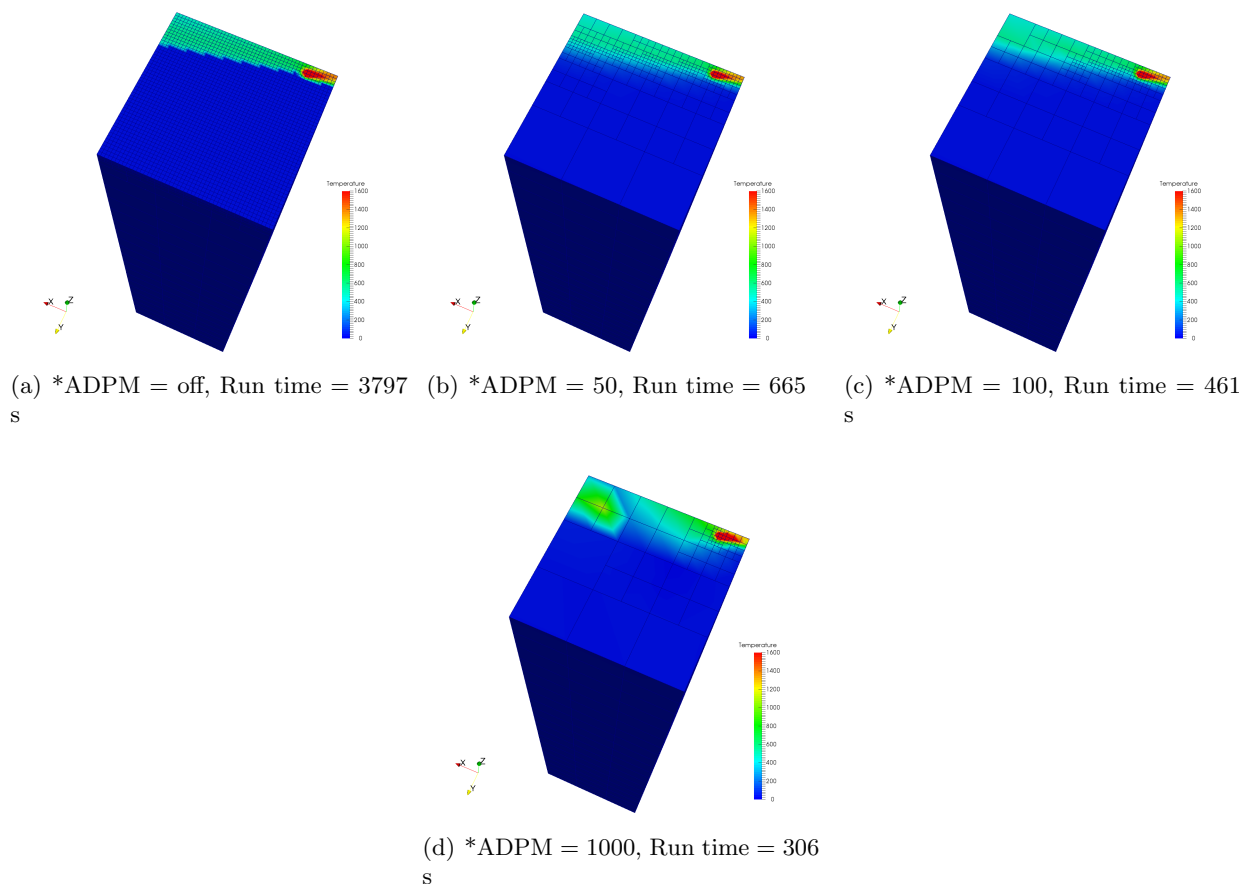


Figure 3.5: Effect of using *ADPM

*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

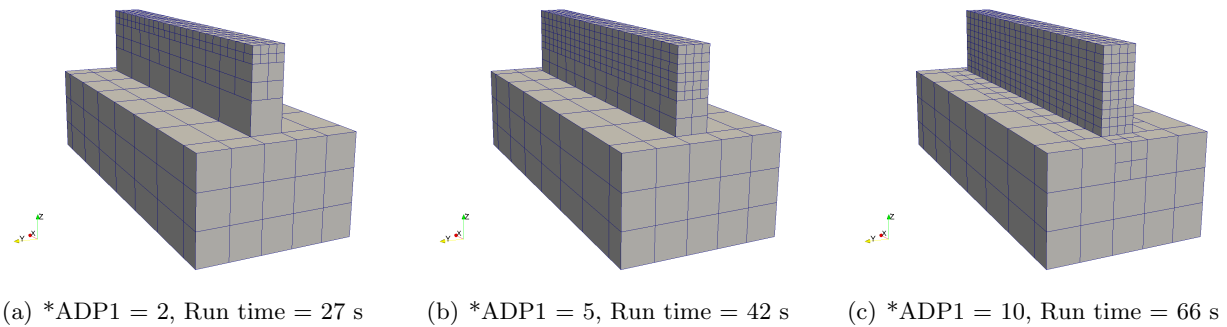


Figure 3.6: Effect of using *ADP1 on the mesh and single CPU run time

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.

*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 [*ADAP](#) mechanical analyses. The stl file must already have been sectioned prior to importing for use in conjunction with the [*SYMM](#) card.

Required Cards: [*ADAP](#)

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.

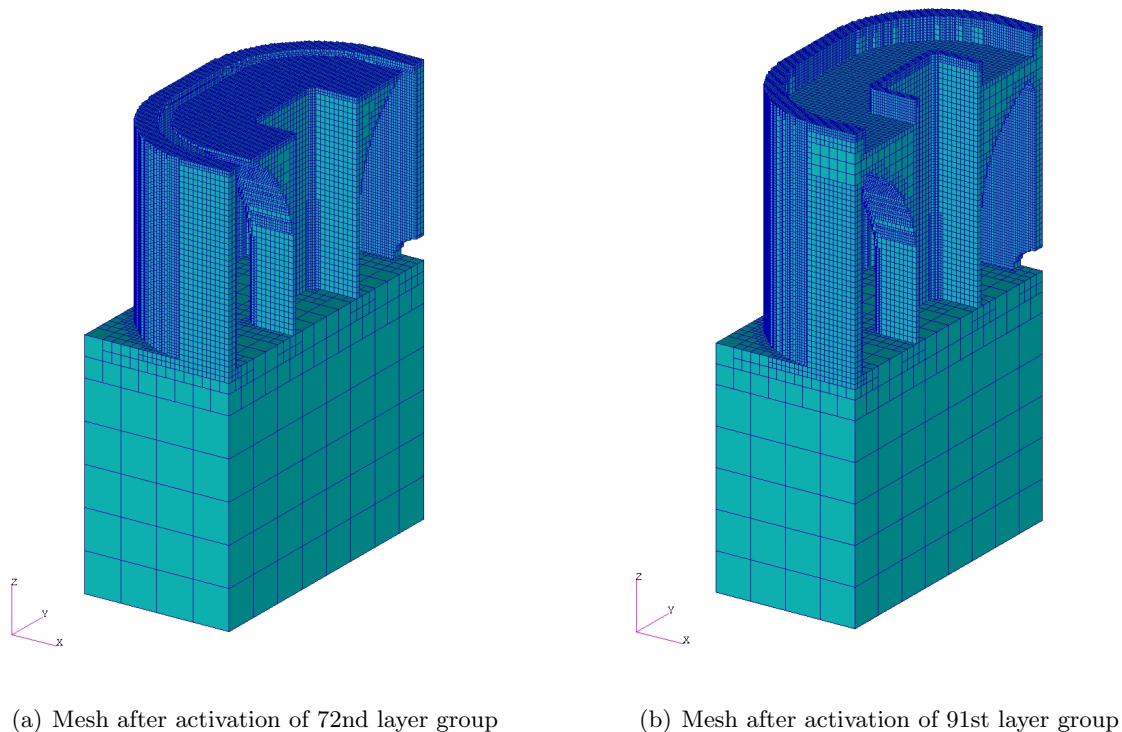


Figure 3.7: Part-Level Powder-Bed Modeling.

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 `*PSSH` 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

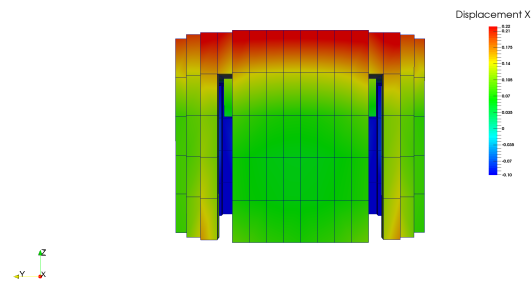
il: i*4: number of layers grouped per element.

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. By default, the minimum ***PBPA** value allowed is 5. This may be altered using ***MNL**.

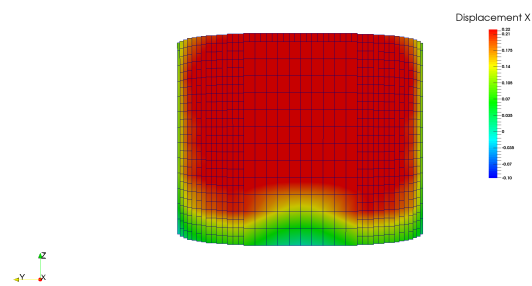
Best Practices: $il*t$ should never be larger than half of the width of the thinnest wall in the structure being modeled. However, this is not always feasible. Using the GUI to generate input files will perform these calculations automatically. Using larger values of ***PBPA** will dramatically increase 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.

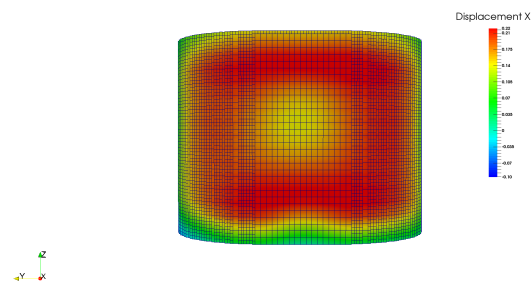
One can see in Figure 3.8(a) that using ***PBPA** values that are too high will not preserve the geometry of the reference stl file. As the ***PBPA** values are decreased geometry (Figure 3.8(b)) is preserved, but the mesh may be too coarse to resolve the solution accurately. As ***PBPA** moves towards 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.



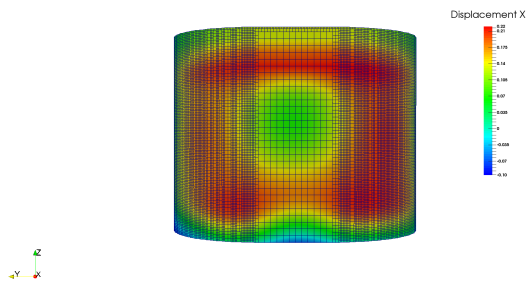
(a) *PBPA=125, Thermo-mechanical run time = 3.12 s



(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.

***MNL: MiNimum LayeR grouping**

*MNL

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 *MNL = 5. Use the *MNL 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 $0.01 \times$ solid part conductivity, and powder specific heat is $0.60 \times$ 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.

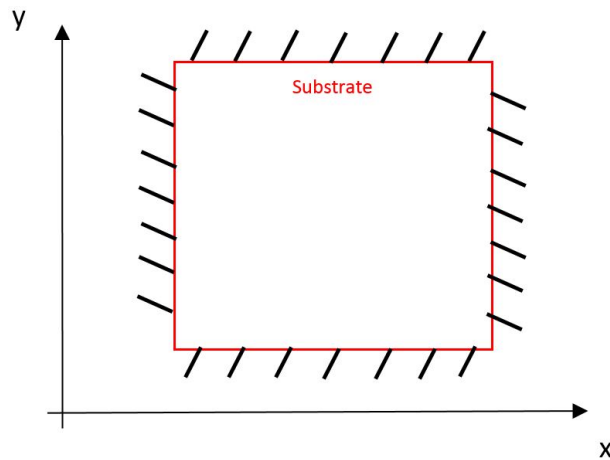


Figure 3.9: Surfaces insulated by ***PBIS**

***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

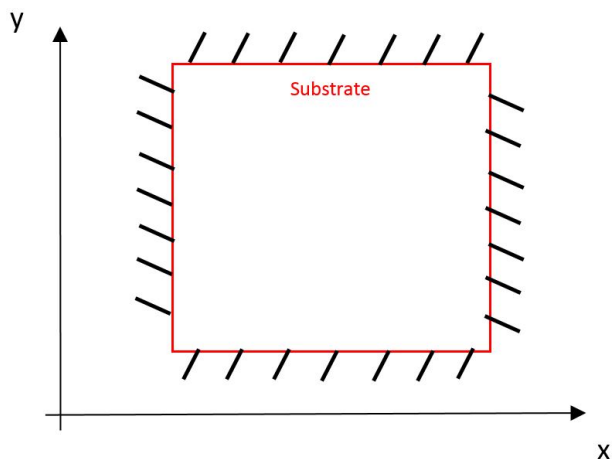
i1

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 (displacement only) 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.

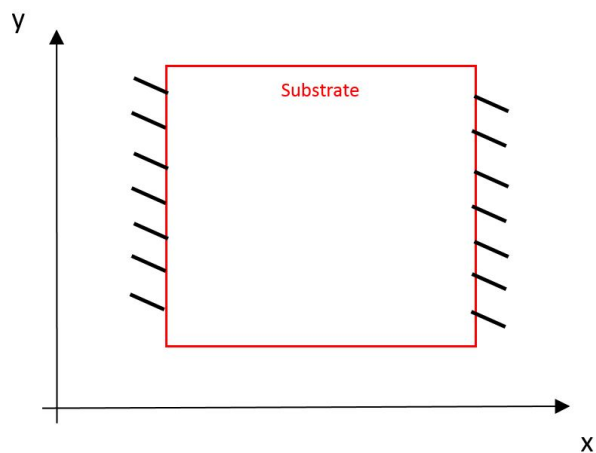
Figure 3.10: Surfaces constrained by `*PBSS`

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.

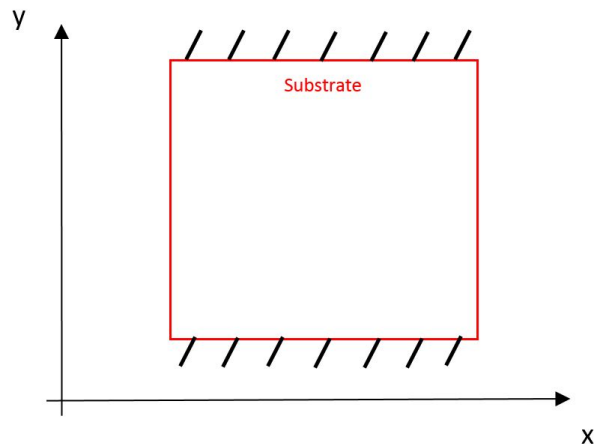


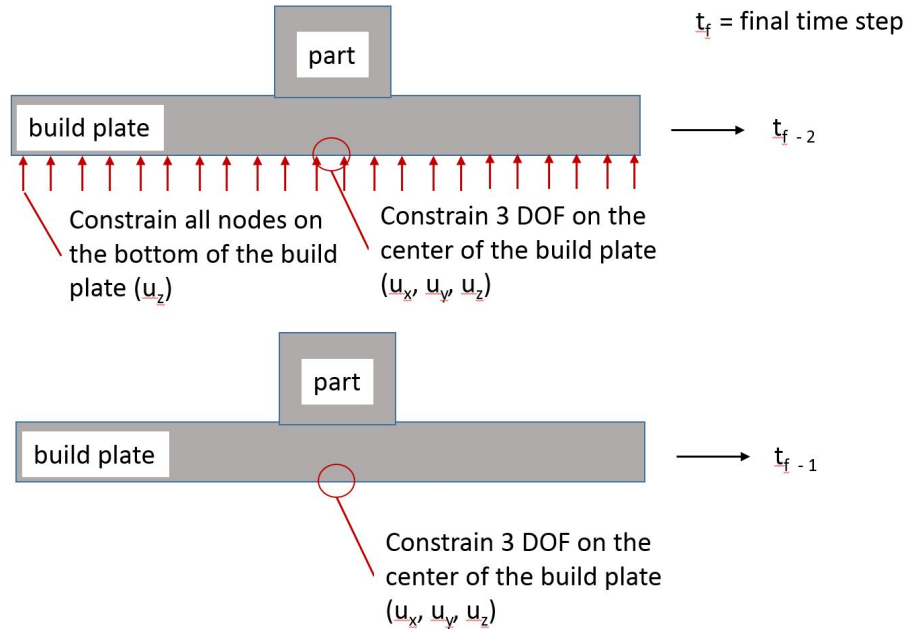
Figure 3.12: BCs assigned by ***PBSY**

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 y 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.

Figure 3.13: BCs assigned by ***FSUB*****FSBT: Floating SuBstrate Type**

***FSBT**
i1

***FSBT** 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 fixed 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.

3.6.1 *NTFT***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 mm³. 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, 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](#).

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](#)

***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: UNIO_n 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

***OSTL: Old STL algorithm switch**

*OSTL

The option will revert the auto mesh algorithm used in before the currently implemented method.

***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 interference of 85% recoater interface. At 0.321 mm build interface, there will be 19% recoater tolerance and a 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 a warning and a calculated interference of -25%, which is depicted in Figure 3.14(d).

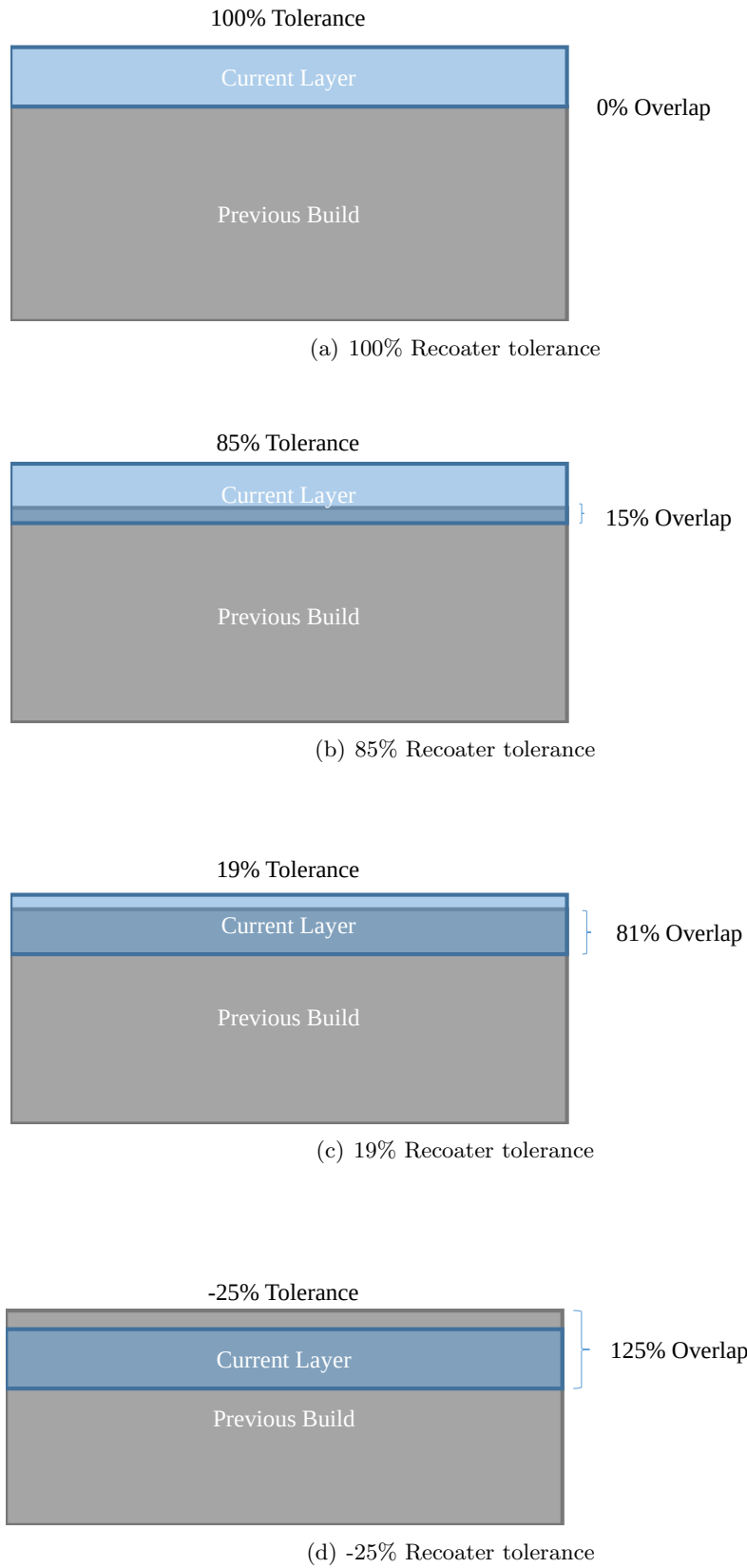


Figure 3.14: Recoater tolerance calculation examples

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
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:

$$t = \frac{V}{vhd}r1 \quad (3.1)$$

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**

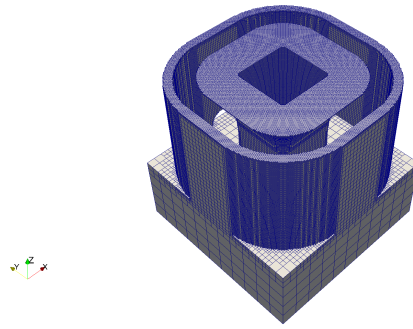
*PBLR

i1

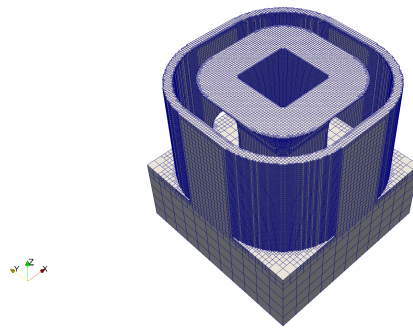
i1: i*4: number of generations coarsened per activation layer. Default 0.

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.

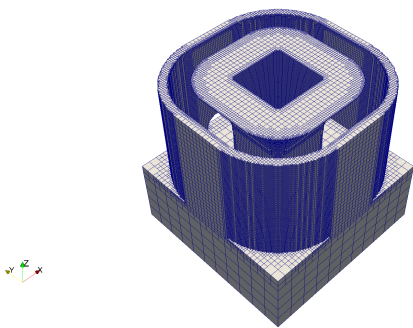
Best Practices: `*PBLR = 1` should achieve a fair compromise between speed and accuracy for most geometries. Larger geometries should use values of `*PBLR = 2`, extremely large builds, `*PBLR = 3`. When performing quantitative stress via `*PPLA` or heat treatment simulations using `*STRF`, lower values 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 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](#). [*DSP](#) 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/mm³.

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.
- r_{12} , r_{22} , r_{32} ,... 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 ***MATID** 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.

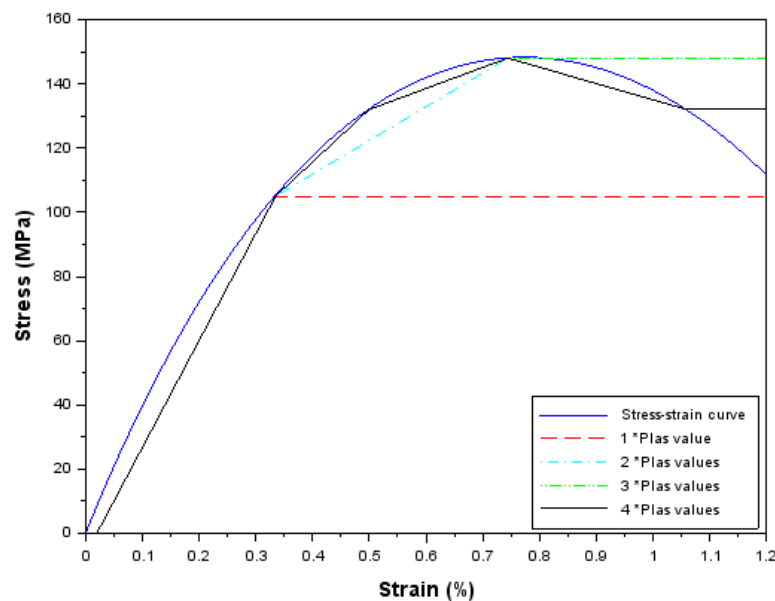


Figure 3.16: Approximating the stress-strain curve using 1-4 values in the ***PLAS** card

***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

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

- 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.

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 temperature, 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 $\text{W/mm}^2 \text{ } ^\circ\text{C}$ 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).

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 3.17(a) and Figure 3.17(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 overrides any **CONV* value set. Consult Figure 3.17(a) and Figure 3.17(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 overrides any **CONV* value set. Consult Figure 3.17(a) and Figure 3.17(b) to see how this card is applied for the powder and no powder cases.

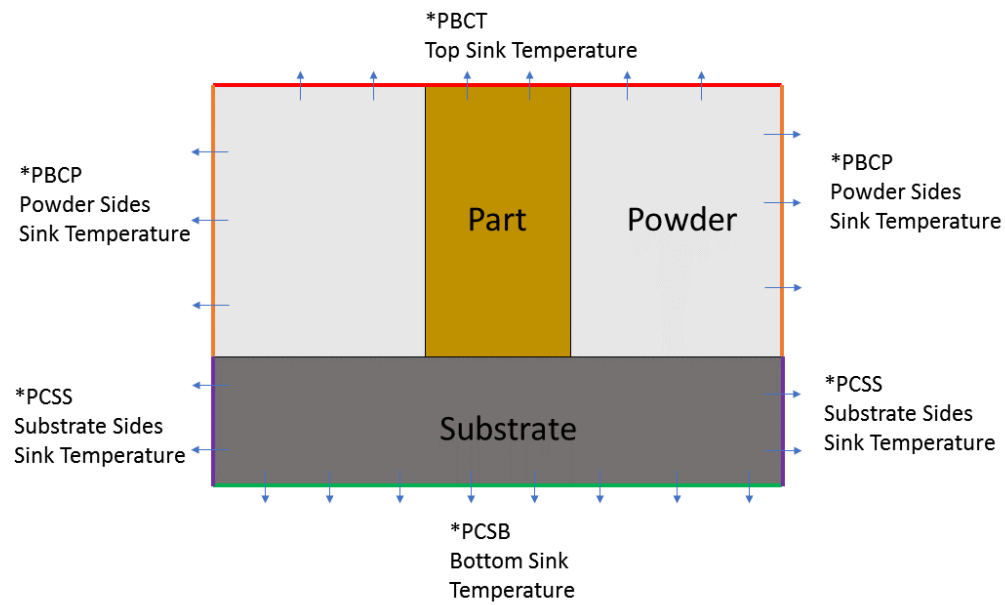
***PBSB: Powder Bed Substrate Bottom convection**

*PBSB

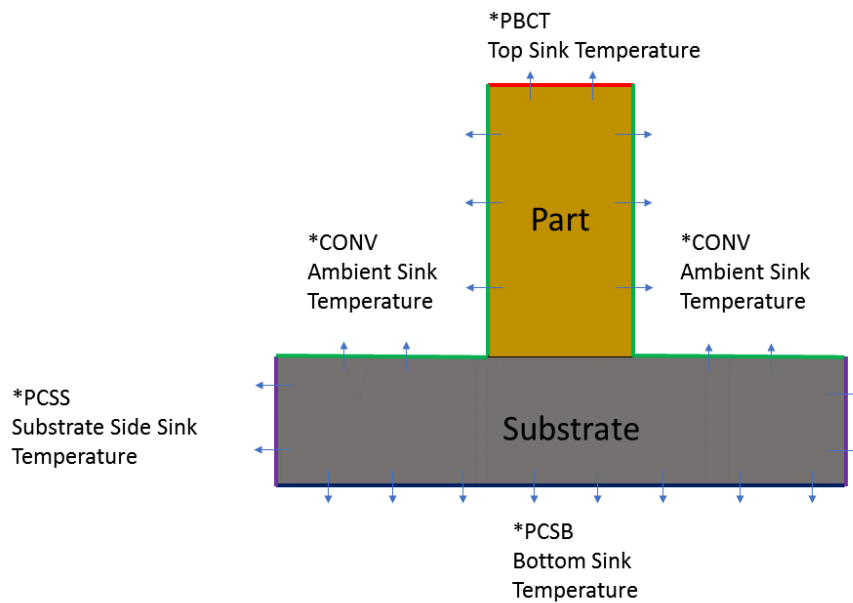
r1, r2

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 overrides any *CONV value set. Consult Figure 3.17(a) and Figure 3.17(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 3.17: 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 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/mm², 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 3.18.

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 $r4*a$

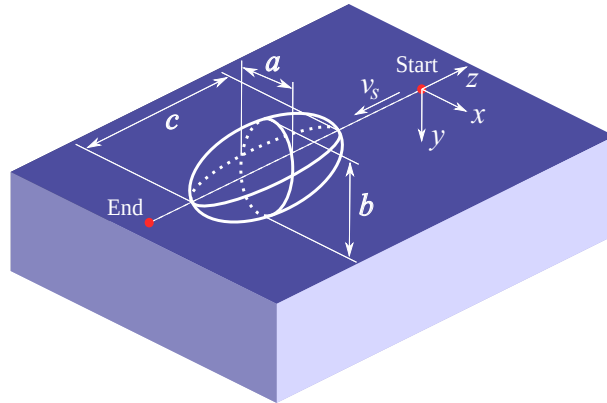


Figure 3.18: Schematic of the Goldak heat source

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.

$$r2 = \text{layer height} / \text{laser radius}$$

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.

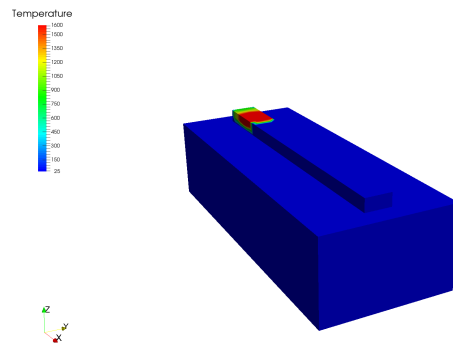
***LINQ: Line heat input (Q) model**

*LINQ

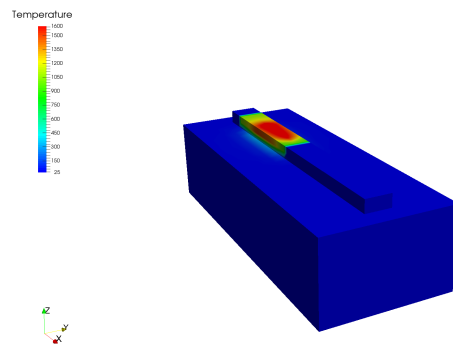
This option activates a line heat input model using an averaged heat source over a line. The length of the line is determined by the *TAUT card. For example, if for a value of 10 is specified by *TAUT, the length of the line is going to be 10 times the radius of the heat source as defined by the *LSRF 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 3.19 for the same simulation without using *LINQ (Figure 3.19(a)).

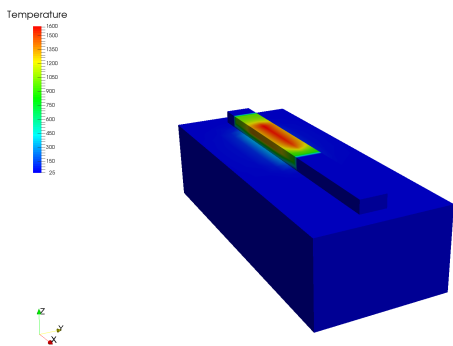
Required Cards: *GOLD, *TAUT, *LSRP or *LSRF.



(a) *LINQ = off



(b) *LINQ = 5



(c) *LINQ = 10

Figure 3.19: 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

r8: r*8: vector angle change from layer to layer

r9: r*8: x width of part

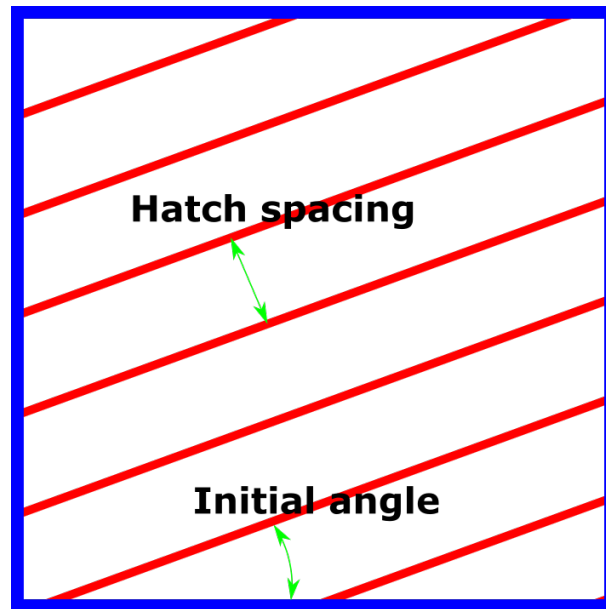
r10: r*8: y width of part (if omitted set to r9)

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).

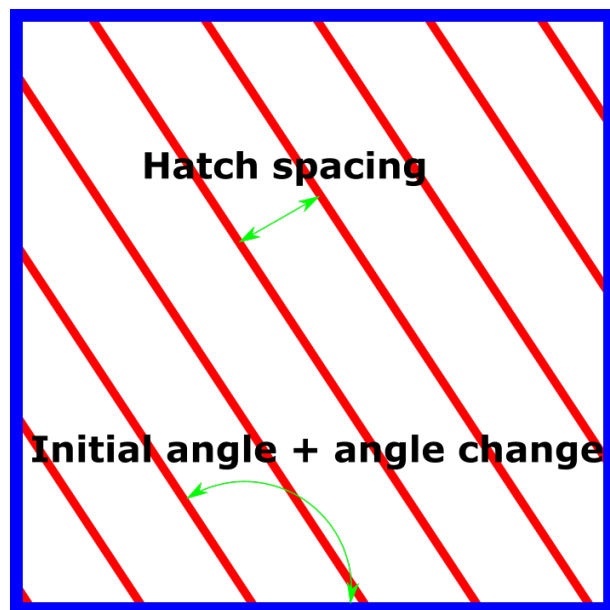
Typically the radius of the melt pool (r2) is about twice the radius of the laser beam.

Figure 3.20 depicts an example schematic of the first two build layers using paths generated using

*LSRP.



(a) Layer 1



(b) Layer 2

Figure 3.20: Illustration of two build layers automatically generated paths using *LSRP1

LSRF: Moving heat source definition file**LSRF**

a1

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.

3.8.4 Mechanical Constraints***NBLT: No BoLTs**

*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 3.21 illustrates the option.

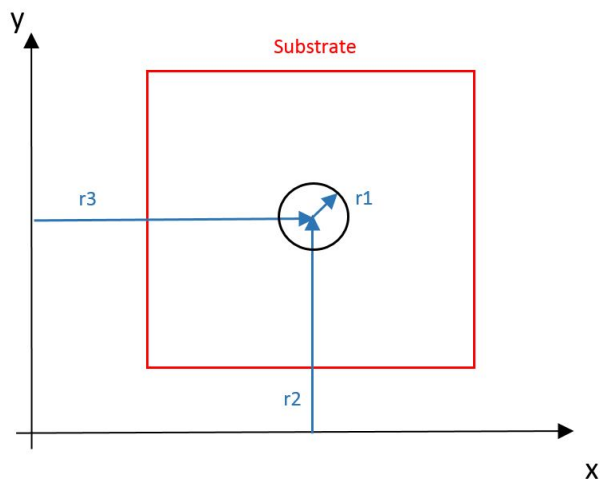


Figure 3.21: Positioning of a circular constraint

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 3.22 illustrates the option.

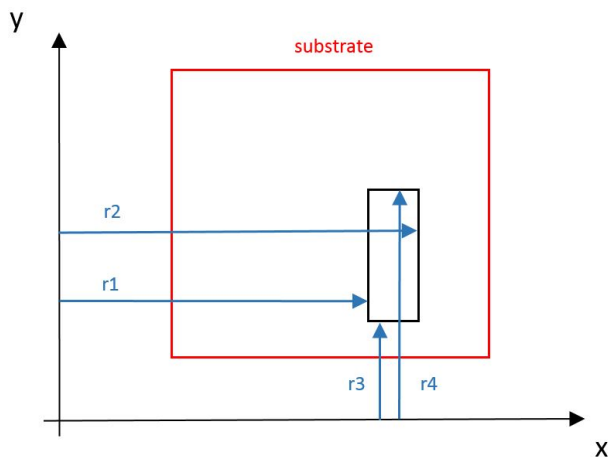


Figure 3.22: Positioning of a rectangular constraint

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 [*FIR](#) 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 [*FIR](#) 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.

***FISR: FItture 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

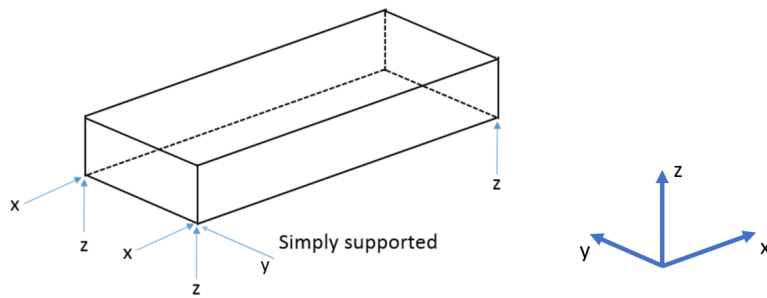
This card is similar to [*FIR](#). However, instead of fixing degrees of freedom, it adds nonlinear springs at specified degrees of freedom. The stiffness of the spring is defined using the [*DASH](#) card.

***FSRD: FItture Spring Rectangular Death**

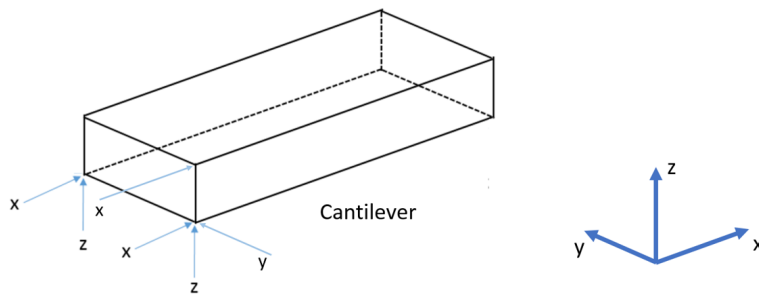
[*FSRD](#)

When this option is used, the springs created with [*FISR](#) get removed when the substrate is removed from machine.

Required cards:[*FISR](#)



(a) *SBBC 1: Simply supported substrate



(b) *SBBC 2: Cantilevered substrate

Figure 3.23: Automatic DED Substrate Boundary conditions assigned by *SBBC

***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 3.23. 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.

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.

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 [*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**

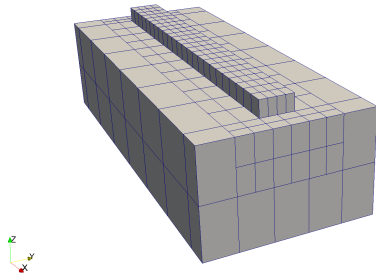
***NELR**

r1

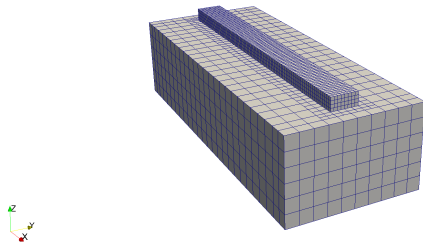
r1: r1*4: Number of element per melt pool radius. Default 1.

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.

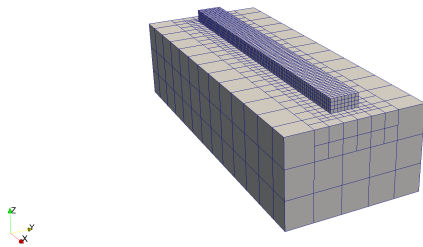
Figure [3.24](#) illustrates the use of the [*NELR](#) card. Note that the change in elements per laser radius will scale up the total number of elements in the model. It is recommended to increase the [*ADAP](#) value to inhibit extraneous element production.



(a) $NELR = 2$, $*ADAP = 2$



(b) $NELR = 4$, $*ADAP = 2$



(c) $NELR = 4$, $*ADAP = 3$

Figure 3.24: 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 3.25 illustrates the option.

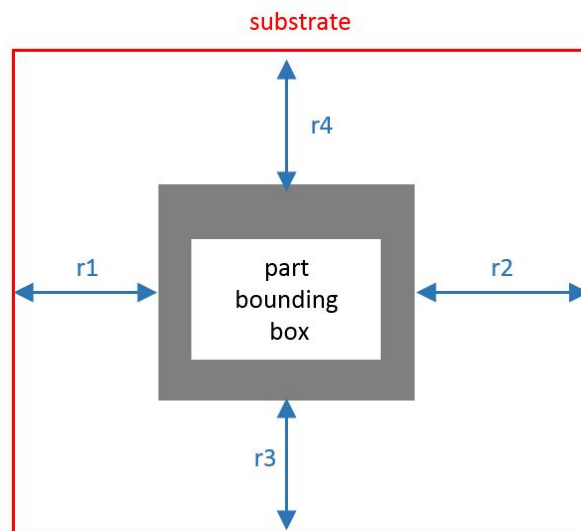


Figure 3.25: Assigning substrate geometry using ***SBXY**

***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 3.26 illustrates the option.

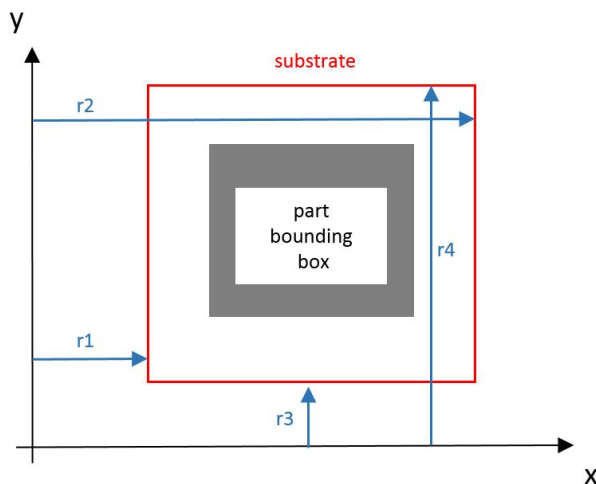


Figure 3.26: Assigning substrate geometry using ***SBDM**

***BOXM: Bounding BoX Mesh output**

*BOXM

This card turns on write output of bounding box base meshes (showing the size of the substrate prior to deposition) for moving source analyses with auto mesh generation, to a file called file_name_geom.case in the simulation directory, where file_name is the extensionless input file name.

***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

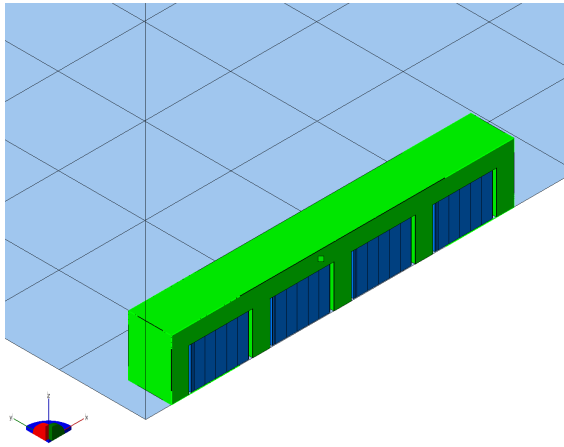
*CLIF

a1

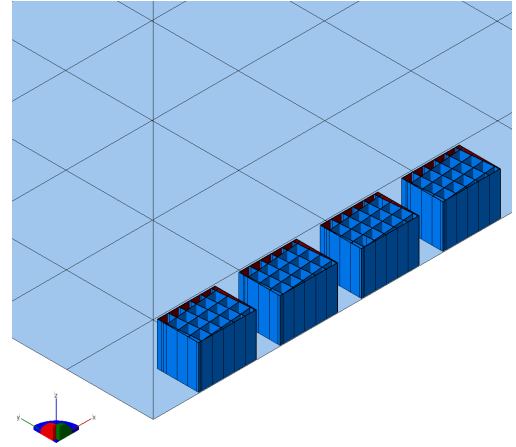
a1: a80: Name of CLI file

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 3.27 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`

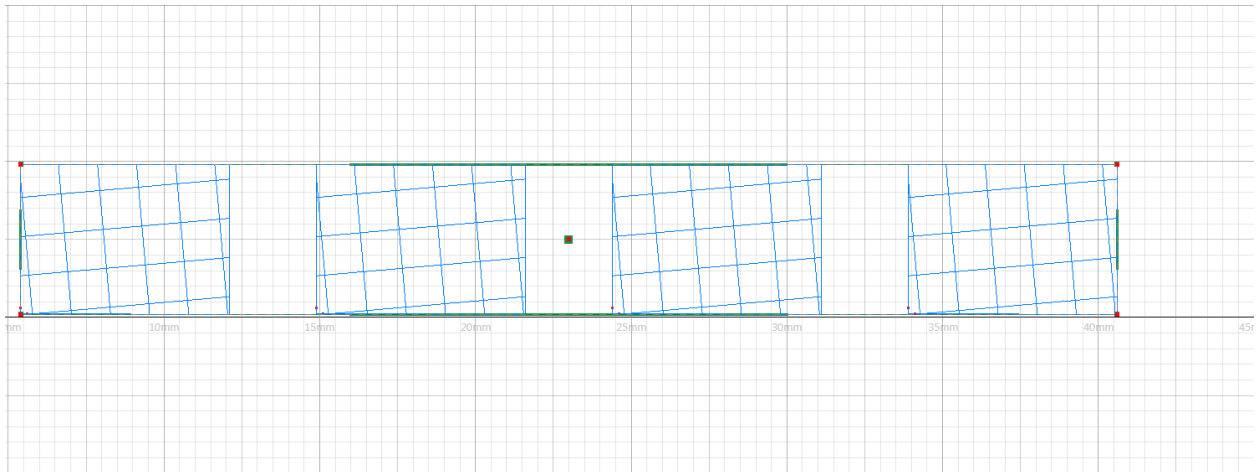
Best Practices: STL based supports are simpler to produce and have smaller file sizes which makes them preferable. It is suggested that *STLF and *STLM be used to import STL type support structures and assign material property IDs and volume fractions.



(a) Example box type supports on STL



(b) Example box type supports



(c) Example box type layer slice

Figure 3.27: Example box type supports

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 Z component of stress at the support-build interface exceeds the specified ultimate tensile stress, the element fails, producing a warning in the output file. A failed element remains in the mesh but does not resist deformation. An example of failed elements is shown in Figure [3.28](#).

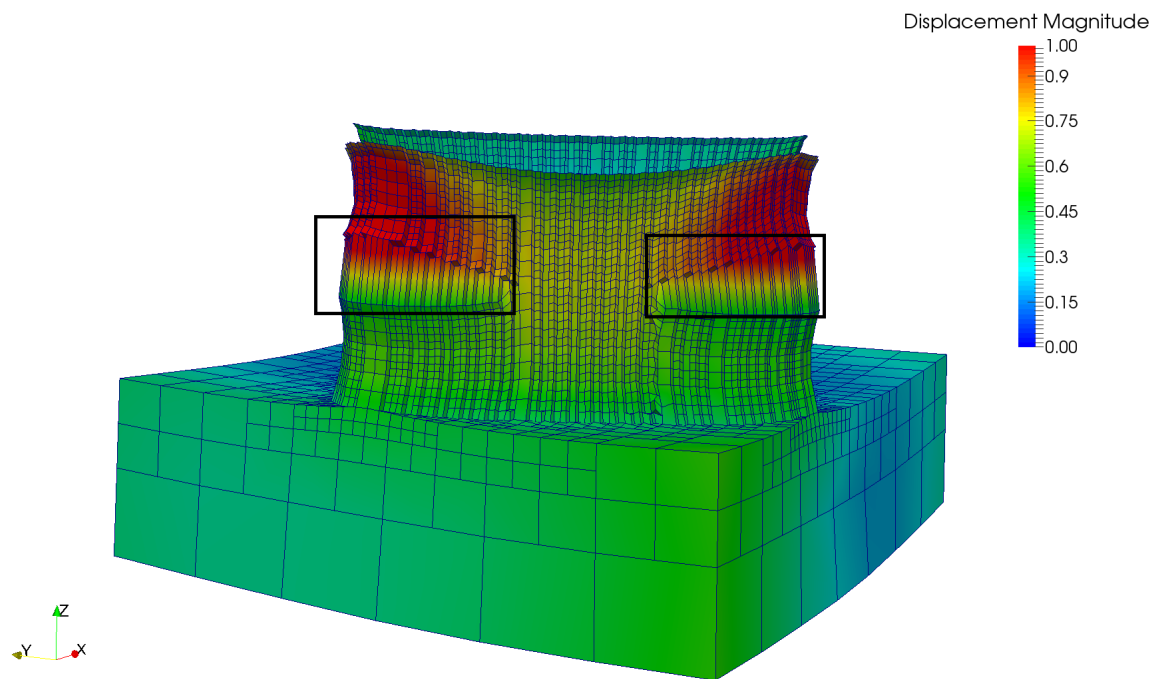


Figure 3.28: Example of failed elements using `*UTSR`, 5X magnification

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](#).

***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. 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 [*MATID](#). 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.

***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/mm²/K.

Required Cards: [*STRF](#), [*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 interrogate the completed results to get thermal or distortion histories at specified points. For points not associated with modal 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

...

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

```

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

```

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. 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 [$\langle \text{ninc} \rangle$, -1] or [1, $\langle \text{ninc} \rangle$], where $\langle \text{ninc} \rangle$ 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 $\langle \text{*INPU} \rangle _ \langle i \rangle _ \text{warp.stl}$, where $\langle i \rangle$ 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, $\langle \text{nnode} \rangle$], where $\langle \text{nnode} \rangle$ 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 $(0, \infty)$

***MAXR : Maximum number of refinement levels**

i1: maximum refinement levels

Default value 10. Range $[1, \infty)$

***MINE : Minimum edge length**

r1: min length

Default value 0.5. Range $(0, \infty)$

***MAXE : Maximum edge length**

r1: max length

Default value 3.0. Range $[<*MINE>, \infty)$

***RTOL : Refinement tolerance**

Default value 1e-3. Range $[0, \infty)$

***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 $<nstl>$.

***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.

***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).

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`

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/(mm² °C)

Density kg/mm³

Displacement mm

Elastic (Young's) Modulus N/mm²

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 $\mu\text{m} / (\mu\text{m } ^\circ\text{C})$ (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 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. Either perform a cloud solve using cloud credits or consider upgrading to Netfabb Local

Simulation.

Number of layers grouped per element *PBPA exceeds max limit 20. Either perform a cloud solve using cloud credits or 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: projectpansupport@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. Either perform a cloud solve using cloud credits, coarsen the mesh, or upgrade to Local Simulation.

Nodes exceed max limit 500000. Either perform a cloud solve using cloud credits, 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. Either perform a cloud solve using cloud credits or 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 [prmggen](#) 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 [prmggen](#) 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 [prmggen](#) 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 [prmggen](#)

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.

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.

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 *APDM 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 ##.

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.

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.

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 *TRANS card, the solver will abort.

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 *LASR or *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 *LSRF 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.

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 available 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.

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 PBIS

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 preceded 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 either `*LASR` or `*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

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-CONTINUOUS 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-CONTINUOUS 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.

Resetting **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.

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.

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, 56
 *3PLP, 88
 *ACNV, 108
 *ADAP, 47
 *ADDP, 33
 *ADP1, 50
 *ADP2, 51
 *ADP3, 51
 *ADPM, 49
 *AMBI, 82
 *ANNL, 79
 *ANTP, 38
 *AUBC, 80
 *AUTM, 99
 *AXSP, 38
 *BINA, 30
 *BMUL, 65
 *BOXM, 102
 *BTOL, 65
 *CAS1, 34
 *CLIF, 103
 *CMSH, 38
 *COND, 73
 *CONV, 83
 *COOL, 98
 *DASH, 74
 *DDM!, 45
 *DDM1, 46
 *DDMC, 46
 *DDMM, 46
 *DDMP, 47
 *DDMZ, 47
 *DENS, 74
 *DEPE, 31
 *DSHP, 74
 *ELAS, 75
 *EMIS, 75
 *EMUL, 65
 *END, 39
 *ENSA, 33
 *ENSF, 34
 *ENSI, 33
 *ETOL, 65
 *EVAP, 79
 *EXPA, 75
 *FINT, 82
 *FISR, 97
 *FIXC, 94
 *FIXR, 95
 *FLxC, 95
 *FLxR, 96
 *FSBT, 61
 *FSRD, 97
 *FSUB, 60
 *GOLD, 87
 *GTAB, 72
 *GTOL, 91
 *HMAT, 72
 *HPRM, 72
 *INIT, 82
 *INPU, 30
 *INTA, 36
 *IOBN, 29
 *KTMP, 40
 *LATE, 76
 *LENS, 87
 *LFUS, 34
 *LINQ, 89
 *LSRF, 94
 *LSRP, 91
 *MATE, 73
 *MATI, 73
 *MLTT, 78
 *MNLr, 56
 *NAPL, 72
 *NELN, 102
 *NELR, 99
 *NLTL, 39
 *NMAT, 80
 *NMTT, 80
 *NOFC, 39
 *NTFE, 62
 *NTFT, 62
 *OFC1, 39
 *OFC2, 40
 *OFC3, 40
 *OFLX, 33
 *OFNS, 31
 *OOPP, 32
 *ORCS, 68
 *ORES, 32
 *OSD3, 36

*OSIG, [36](#)
*OSP3, [36](#)
*OSTL, [66](#)
*OSVM, [36](#)
*OWFC, [32](#)
*PARL, [101](#)
*PBCH, [69](#)
*PBCP, [84](#)
*PBCT, [83](#)
*PBDL, [69](#)
*PBIS, [57](#)
*PBLR, [69](#)
*PBPA, [54](#)
*PBPF, [56](#)
*PBS2, [58](#)
*PBSB, [84](#)
*PBSH, [57](#)
*PBSN, [58](#)
*PBSS, [58](#)
*PBSX, [59](#)
*PBSY, [60](#)
*PCSS, [84](#)
*PLAS, [76](#)
*PPLA, [37](#)
*RAMP, [81](#)
*RCTR, [66](#)
*RELA, [40](#)
*REST, [31](#)
*RSTL, [66](#)
*SBCC, [98](#)
*SBDM, [102](#)
*SBXY, [101](#)
*SOLU, [41](#)
*SPA8, [41](#)

*SPEC, [78](#)
*SRLX, [78](#)
*STL1, [64](#)
*STL2, [64](#)
*STLC, [63](#)
*STLF, [62](#)
*STLM, [63](#)
*STLS, [65](#)
*STOL, [64](#)
*STRF, [107](#)
*STRR, [107](#)
*SUB2, [52](#)
*SYMM, [52](#)
*TAMB, [83](#)
*TAUT, [43](#)
*TCUT, [79](#)
*TITL, [30](#)
*TPRE, [35](#)
*TPRM, [43](#)
*TRAN, [42](#)
*TWAL, [44](#)
*UNIO, [64](#)
*UTSR, [105](#)
*VCLI, [105](#)
*WRTU, [33](#)
*ZDWL, [43](#)

Execution and File Notation, [21](#)

Getting Started, [7](#)

Keyword Listing, [29](#)

prngen, [24](#)