NORTHWESTERN
UNIVERSITY

How to Use NU AM-CFD

Kevontrez Jones

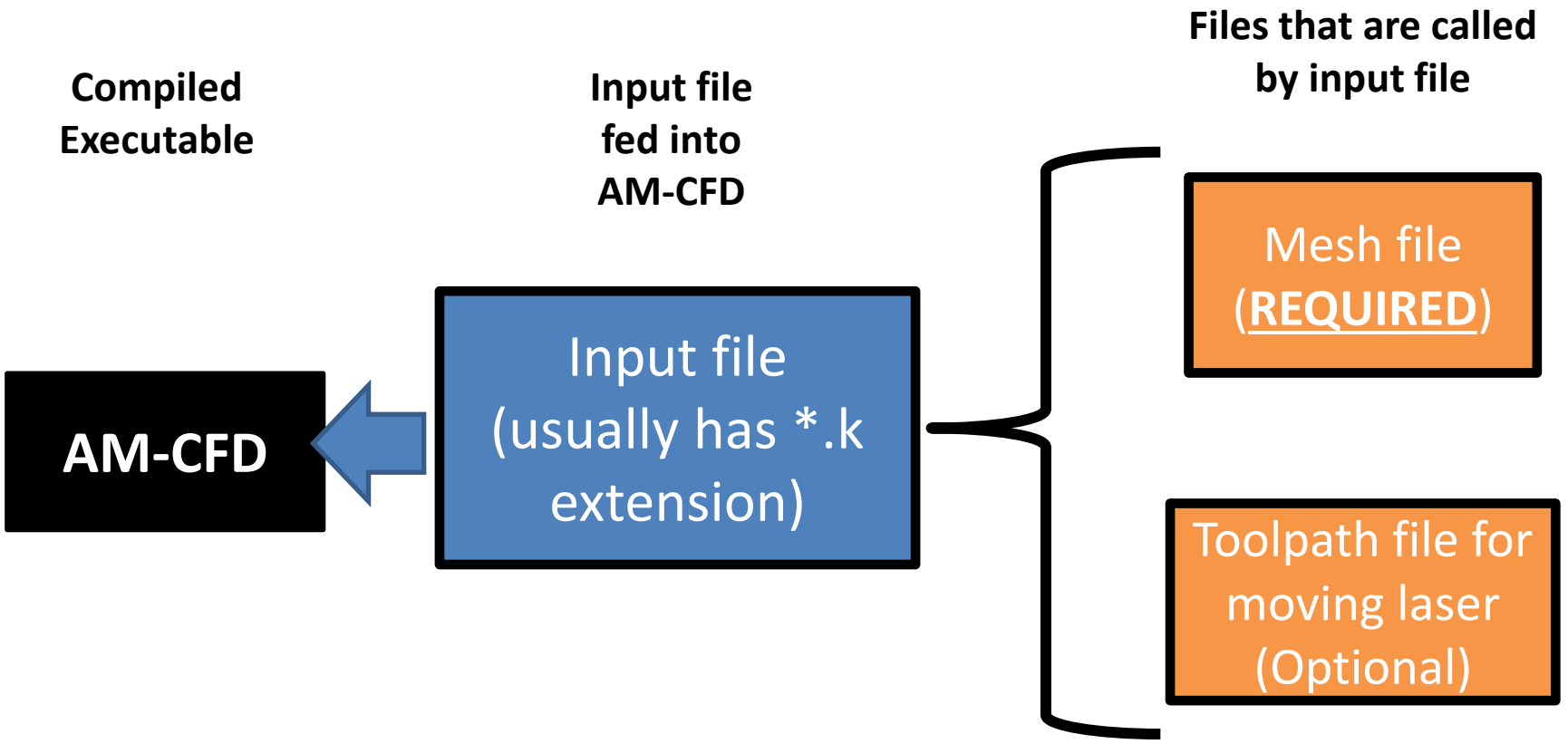
Some Things To Know

- AM-CFD reads input files/mesh files similar (nearly identical) to a Keyword format from LS-DYNA
- Throughout this tutorial, examples are given from functioning input files.
- Lines that start with "$" indicate a comment and are automatically ignored by the CFD software.
- The input and mesh file uses are not written in any specific order. This means that you can rearrange the keywords however you like, and it will not matter. **HOWEVER, IT IS IMPORTANT THAT THE CORRESPONDING PARAMETERS FOR EACH KEYWORD ARE WRITTEN IN THE CORRECT ORDER**
- To visualize the full-field results, an external post-processor that can read the VTK file formats is needed. ParaView, an open-source software, has been used in our research group, but other options such as Visit exist.

CFD Code Requires an Input File to Run

An input file is fed into the CFD software to run the simulations

- This input file **DOES NOT** contain the mesh itself
- This input file will call other files **BY LOCATION AND NAME** required for the simulation
- The Input file contains other parameters (discussed later)



Contents of the Mesh File

*X_Grid_PARAMETERS

- Defines the mesh in the x-direction

*Y_Grid_PARAMETERS

- Defines the mesh in the y-direction

*Z_Grid_PARAMETERS

- Defines the mesh in the z-direction

EXAMPLE:

2		# of mesh partitions in the desired direction
1.02e-3	0.38e-3	Length of each mesh partition
34	76	# of elements in each partition
-1.5	1.0	Coefficients to describe transition between partitions

Contents of the Toolpath File

- This file is used for the moving laser flux boundary condition
- Lists the (x, y, z) position of the laser at specific time states as well as the state of the laser
- Laser state is: 0 = off, 1 = on
- Current laser state is based on the laser state that is at a time greater than or equal to the current time (looks ahead for information)
- Will linearly interpolate toolpath position
- **UNITS OF TOOLPATH FILE MUST BE CONSISTENT WITH THE MESH**

NORTHWESTERN UNIVERSITY

Contents of the Toolpath File

- From lines 1-2, laser is off
- At line 3, laser finally turns on. For this example, if the current time is 0.2, then the laser is assumed to be on and will linearly interpolate position from lines 2 and 3
- At line 4, laser turns off. For this example, if the current time is 0.6, then the laser will be off.
- At line 5, laser turns on again. Rinse and repeat

EXAMPLE:

	Time	X-position	Y-position	Z-position	State
1	0.00000000	0.00000000	0.00000000	0.00000000	0
2	0.00000000	0.00000000	0.00000000	0.80000001	0
3	0.40000000	4.00000000	0.00000000	0.80000000	1
4	0.90000000	4.00000000	0.00000000	0.80000000	0
5	0.98000000	4.00000000	0.80000001	0.80000000	1
6	1.38000000	0.00000000	0.80000001	0.80000000	1
7	1.88000000	0.00000000	0.80000001	0.80000000	0
8	1.96000000	0.00000000	1.60000002	0.80000000	1
9	2.36000000	4.00000000	1.60000002	0.80000000	1
10	2.86000000	4.00000000	1.60000002	0.80000000	0

NORTHWESTERN UNIVERSITY

How are Files Called Through the Input File?

- Files are called through specific keywords (identical to LS-DYNA keyword formatting)
- For files, need to specify the name of the file as well as its path if its not in the same directory.

EXAMPLE:

```
*MESH_FILE
../../MeshFiles/multiTrackmesh_01x01.k

*TOOLPATH_FILE
../../ToolFiles/MultiLayerTest.crs

…

…

…

…

…
```

- *MESH_FILE Defines the name of the mesh file
- *TOOLPATH_FILE defines the toolpath file
- **IMPORTANT: In this example, it is assumed that the input file is in a sub-level directory, and there are separate sub-directories called “ToolFiles” and “MeshFiles” that contain necessary files**

Alternative Toolpath Keyword (Single-track)

- *SINGLE_TRACK_TOOL is an optional means to define a toolpath for a single-track simulation (with or without powder)

EXAMPLE:

```
*SINGLE_TRACK_TOOL
$ lasertime  xstart     ystart     vel
  1.000      0.500e-3   1.000e-3   1.5
```

About of time for laser to be "on"

Starting x-position of laser head

Starting y-position of laser head

Scan speed of the laser

NORTHWESTERN UNIVERSITY

Defining Output Files for Visualization

- *KEYWORD_ID defines the name of the output file for visualization
  - The file to load in to ParaView will end with a "*.pvd" extension. For this example, you will need to load "my-output-file.pvd".
- *DIRECTORY names the directory for which the individual output files will for each time step will be store. In this example, a directory called "some_directory" will be created in the directory you ran the simulation.
  - If this keyword is left out, the default directory name will be "Result".

```
*KEYWORD_ID
my-output-file
*DIRECTORY
some_directory
```

Defining Thermal Material Properties

- Based on prior experience, the standard way to inputting material properties is a mix of constant values and temperature-dependent polynomial expressions

- Material properties for the solid and liquid phases are distinguished from each other

- Linear polynomials:

$$\varphi = aT + b$$

- Quadratic polynomials:

$$\theta = aT^2 + bT + c$$

Material Properties

```
*MATERIAL_PROPERTIES
$ dens   denl    viscos   tsolid   tliquid   hsmelt   hlfriz
  8440   7640    0.007    1563     1623      861e3    1151e3
$ acpa     acpb     acpl
  0.2441   338.59   709.25
$ thconsa  thconsb   thconsc  thconla thconlb
  0.0      0.0155    5.0435   0.0     30.078
```

Order: left to right, top to bottom:

THE SPECIFIC ORDER MUST BE FOLLOWED, BUT ANY NUMBER OF LINES MAY BE USED

- dens = Density of the solid phase
- denl = Density of the liquid phase
- viscos = Dynamic viscosity of the liquid phase
- tsolid = Solidus temperature
- tliquid = Liquidus temperature
- hsmelt = Solidus enthalpy
- hlfriz = Liqidus enthalpy
- acpa, acpb = Linear polynomial coefficients for specific heat of the solid phase
- acpl = Specific heat of the liquid phase
- thconsa, thconsb, thconsc = Quadratic polynomial coefficients for thermal conductivity of the solid phase
- thconla, thconlb= Linear polynomial coefficients for thermal conductivity of the liquid phase

Defining Nodal Outputs

Nodal outputs are defined as *SCALAR_OUT (one degree of freedom per node) or *VECTOR_OUT (three degrees of freedom per node)

EXAMPLE:

```
*SCALAR_OUT
temp
uvel
vvel
wvel
solfrac
csfrac
rho
diff
```

IMPORTANT: VISUALIZATION OF OUTPUTS IS DONE IN EXTERNALLY

NORTHWESTERN UNIVERSITY

Optional Keywords for Output

- *OUTPUT_SECTION is an optional output for writing out a small prism section of data at a higher frequency than typically used for visualization
  - This is used in conjunction with the 3D CA microstructure code (external software)
  - Currently the CA software is being managed by Dr. Greg Wagner's group

```
*OUTPUT_SECTION
$ gxmin    gymin         gzmin
  2.5e-3   0.4175e-3     0.4300e-3
$ gxmax    gymax         gzmax
  3.5e-3   0.5825e-3     0.9000e-3
$ timeout
  2.0e-5
```

(x, y, z) coordinates of lower bounding corner of prism

(x, y, z) coordinates of upper bounding corner of prism

Frequency of output for section data

Defining Time Parameters

- *CONTROL_TERMINATION defines the physical termination time of the simulation (0.67 milliseconds in this example)
  - max_iter = Maximum number of iterations for nonlinear solves per timestep
  - nonlin_tolerance = Tolerance used to stop nonlinear solves per timestep

- *CONTROL_TIMESTEP defines the time increment to take for the simulation (20.0 microseconds)
  - output_timestep = Frequency of outputting data

```
*CONTROL_TERMINATION
$ total_time  max_iter  nonlin_tolerance
  0.00067     100       5.0e-4
*CONTROL_TIMESTEP
$ sim_timestep  output_timestep
  2.0e-5        2.0e-5
```

NORTHWESTERN UNIVERSITY

Constant Parameters For Boundary Conditions

- *PARAMETER defines various parameters to be used by boundary conditions and source terms which are generally unchanged constants
  - BE MINDFUL OF UNITS

```
*PARAMETERS
accelGravity 9.8
radBoltzmann 5.67e-8
ambientTemp  295
ambientPres  101.0e3
gasConstant  8.314
```

← Acceleration of gravity

← Stefan-Boltzmann constant

← Ambient temperature

← Ambient pressure

← Universal gas constant

Options For Thermal Boundary Conditions

- *GAUSS_LASER_PARAMETERS defines parameters to be used by moving laser flux boundary condition using a surface (2D) gaussian distribution

- *VOLUMETRIC_HEAT_SOURCE defines parameters to be used by moving laser flux boundary condition using a volumetric (3D) cylinder with a gaussian distribution

```
*GAUSS_LASER_PARAMETERS
$  power   absorb    radius    factor
   100.0    0.20     40e-6     2


*VOLUMETRIC_HEAT_SOURCE
$  power    apowseta    heatthick   heatrb    factor
   100.0    0.35        75e-6       40e-6     2
```

Laser power

Energy absorption efficiency

Height of cylindrical source

Width of the Gaussian distribution

NORTHWESTERN UNIVERSITY

Options For Thermal Boundary Conditions

- *BOUNDARY_TEMPERATURES defines a constant initial temperature field on the free surfaces and in the body

- *BOUNDARY_ENERGY_EQUATION defines coefficients to account for energy loss due to natural convection and radiation on the free surfaces

```
*BOUNDARY_TEMPERATURES
$ west   east   north   bottom   preheat
  353    353    353     353      353
```

Initial temperatures on free surfaces (+x, -x, +y, -z)

Preheat temperature (body)

```
*BOUNDARY_ENERGY_EQUATION
$ htci   htcj   htckl   htckn   emiss
  10     10     10      10      0.4
```

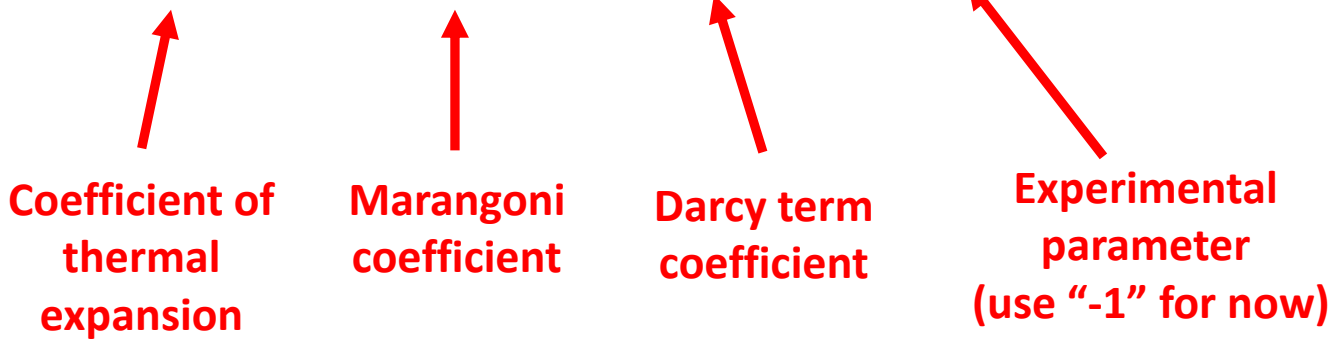
Convection coefficients on free surfaces (+x, -x, +y, -z)

Emissivity due to radiation

Options For Momentum Boundary Conditions

- *PARAMETER defines various parameters to be used by boundary conditions and source terms

```
*BOUNDARY_MOMENTUM_EQUATION
$ beta   dgdtp   darcyKo   a_oxygen
  5e-5   -3.8e-4   1.6e4      -1
```



Options For Numerical Convergence

- *UNDER_RELAXATION defines various parameters to be used for numerical relaxation of solution variables (for convergence)

```
*UNDER_RELAXATION
$ urfu   urfv   urfw   urfp   urfh
  0.8    0.8    0.8    0.8    0.8
```

Relaxation parameters
(u-velocity, v-velocity, w-velocity, pressure, enthalpy)

NORTHWESTERN UNIVERSITY

Options For Powder-bed Simulations

- *POWDER_BED_MODE defines various parameters to be used for powder-bed simulations

```
*POWDER_BED_MODE
$ numlayers  ncvlayer  layerheight  density  cpa     cpb     thcona     thconb
  8          8         0.04e-3      4330     0.2508  357.7   0          0.995
```

Order: left to right:

- numlayers = Number of powder layers
- ncvlayer = Number of control volumes per powder layer
- layerheight = Height of each powder layer
- density = Density of the powder bed
- cpa, cpb = Linear polynomial coefficients for specific heat of the powder state
- thcona, thconb = = Linear polynomial coefficients for thermal conductivity of the powder state