

# **gmd** User's Guide

Tim Fuller

September 2, 2013

# Chapter 1

## Introduction

`gmd` is a Generalized Model Driver.

## Chapter 2

# Building gmd

gmd's code base is largely written in Python and requires no additional compiling. However, the exodusII third party library and material models written in fortran must be built.

### 2.1 Setting Up

Set up and build the third party libraries.

```
% cd GMD/toolset
% python setup.py
```

Add GMD/toolset to PATH.

### 2.2 Building

Build the material libraries

```
% buildmtls
```

## Chapter 3

# Running

Make sure `GMD/toolset` is on your `PATH`.

```
% gmd runid[.xml]
```

The following files will be produced

```
% ls runid.*
```

```
runid.exo      runid.log      runid.xml
```

`runid.exo` is the exodusII output database, `runid.log` the log file, and `runid.xml` the input file.

## Chapter 4

# User Input

User input is via xml control files. In general, tags use CamelCase and attributes lower case. Attributes are described in this document as

`attr="type[default]{choices}"`

where `default` is the default value (if any) and `{choices}` are valid choices (if any). Any attribute not having a default value are required. Types are `str`, `int`, `real`, `list`. Lists are given as space separated lists (i.e., "1 2 3").

### 4.1 GMDSpec

`<GMDSpec>`

All input files must have as their root element `<GMDSpec>`. Recognized subelements of `<GMDSpec>` are

- `<Physics>`
- `<Permutation>`
- `<Optimization>`

Additionally, the following elements are read from anywhere in the input file

- `<Include>`
- `<Function>`
- `<TerminationTime>`

## 4.2 Preprocessing

Preprocessing allows specifying variables in the input inside of comment tags for use in other parts of the input. Syntax mirrors that of **aprepro**.

### 4.2.1 Example

Specify the `<Material>` parameter `K` and `<Path>` parameter `estar` as variables

```
<GMDSpec>
  <!-- {K = 23e9}
        {estar = -.05}
  -->
  <Physics>
    <Material model="elastic">
      <K> {K} </K>
      <G> 54e9 </G>
    </Material>
    <Path type="prdef" estar="{estar}">
      ...
    </Path>
  </Physics>
</GMDSpec>
```

## 4.3 Include

```
<Include href="str"/>
```

Path to file to be included as if its contents were inplace in the input file

### 4.3.1 Example

```
<Include href="/path/to/some/file.ext"/>
```

## 4.4 Function

```
<Function id="int"
  type="str{analytic expression, piecewise linear}"
  var="str[x]" href="str" cols="list[1 2]">
```

Define functions to be used elsewhere in input. `id=0` and `id=1` are reserved for the constant 0 and 1 functions, respectively.

#### 4.4.1 Examples

##### Analytic expression

```
<Function id="2" type="analytic expression" var="t">
    sin(t)
</Function>
```

##### Piecewise linear table

```
<Function id="2" type="piecewise linear">
    1 2
    2 3
    3 5
</Function>
```

Read a piecewise linear table from an external file using columns 1 and 3

```
<Function id="2" type="piecewise linear" href="./file.dat" cols="1 3"/>

% cat file.dat
# Column1 Column2 Column3
1 1 4
2 3 7
.
.
.
100 4.2 1.43
```

## 4.5 TerminationTime

```
<TerminationTime> float </TerminationTime>
```

Termination time for simulation. If not specified, termination time is taken as final time in `<Path>`.

#### 4.5.1 Example

```
<TerminationTime> 1.e-6 </TerminationTime>
```

## 4.6 Physics

```
<Physics driver="str[solid]{solid, eos}">
```

Define the physics of the simulation. Recognized subelements of `<Physics>` are

- `<Path>`
- `<Surface>`
- `<Material>`
- `<Extract>`

### 4.6.1 Path

**Supporting Drivers:** `solid`

```
<Path type="str{prdef}"  
      format="str[default]{default, table, fcnspec}"  
      cols="list[1, ..., n]" cfmt="str" tfmt="time"  
      nfac="int[1]" kappa="real[0]"  
      tstar="real[1]" estar="real[1]" sstar="real[1]"  
      amplitude="real[1]" ratfac="real[1]" href="str">
```

Define deformation paths. The  $j$ th leg of `<Path>` is sent to the driver in form `[tf, n, cfmt, Cij]`, where `tf`, `n`, `cfmt`, and `Cij` are the termination time, number of steps, control format, and control values.

A note on `cfmt` and `Cij`. `cfmt[i]` instructs the driver as to the type of deformation represented by `Cij[i]`. Supported `cfmt` are described in Table 4.1. For example, the following `cfmt` instructs the driver that the components of `Cij` represent [stress, strain, stress rate, strain rate, strain, strain], respectively: `cfmt = 423122`. Mixed modes are allowed only for components of strain rate, strain, stress rate, and stress. Electric field components can be included with any deformation type.

The components `Cij` take the following order

**Vectors:** [X, Y, Z]

**Symmetric tensors:** [XX, YY, ZZ, XY, YZ, XZ]

**Tensors:** [XX, XY, XZ, YX, YY, YZ, ZX, ZY, ZZ]

If `len(Cij)  $\neq$  6` (or 9 for deformation gradient), the missing components are assumed to be zero strain.



cfmt	Deformation type
1	Strain rate
2	Strain
3	Stress rate
4	Stress
5	Deformation gradient
6	Electric field

Table 4.1: Supported deformation types and cfmt code

### Examples

The following examples will help clarify the <Path> input syntax

**format: default** Uniaxial strain, all six components of strain prescribed

```
<Path type="prdef" kappa="0" tstar="1" estar="-.5" amplitude="1" ratfac="1">
  <!-- termination time, number of steps, cfmt, Cij -->
  0 0 222222 0 0 0 0 0
  1 100 222222 1 0 0 0 0
  2 100 222222 2 0 0 0 0
  3 100 222222 1 0 0 0 0
  4 100 222222 0 0 0 0 0
</Path>
```

**format: default** Uniaxial strain, stress controlled

```
<Path type="prdef" nfac="100">
  0 0 444 0 0 0
  1 1 444 -7490645504 -3739707392 -3739707392
  2 1 444 -14981291008 -7479414784 -7479414784
  3 1 444 -7490645504 -3739707392 -3739707392
  4 1 444 0 0 0
</Path>
```

**format: default** Uniaxial stress, mixed mode

```
<Path type="prdef" nfac="100">
  0 0 222 0 0 0
  1 1 244 {epsmax} 0 0
  4 1 244 0 0 0
</Path>
```

**format: table** Read legs from table. Control type is uniform for all legs. Specify control type as `cfmt` attribute of `<Path>`. Optionally, specify the time format as `tfmt` and number of steps for each leg as `nfac`.

```
<Path type="prdef" format="table" cols="1:4" cfmt="222" tfmt="time">
  0 0 0 0
  1 1 0 0
  ...
  n 2 0 0
</Path>
```

**format: table** Read the table from a file, first by the `<Include>` element and then the `href` attribute.

```
<Path type="prdef" format="table" cols="1 3:8" cfmt="222222" tfmt="time">
  <include href="exmpls.tbl"/>
</Path>
```

```
<Path type="prdef" format="table" cols="1 3:8" cfmt="222222" tfmt="time"
  href="exmpls.tbl"/>
```

**format: fcnspec** Create legs from functions. Functions are specified as `function id[:scale]`. Syntax is otherwise similar to table format. Only a single leg can be specified.

```
<Path type="prdef" kappa="0" tstar="1" amplitude="1" format="fcnspec"
  cfmt="222" nfac="200">
  {2 * pi} 2:1.e-1 1:0 1:0
</Path>
```

## 4.6.2 Surface

**Supporting Drivers: eos**

```
<Surface format="str[default]{default, table}"
  cols="list[1, ..., n]" cfmt="str"
  nfac="int[1]" tstar="real[1]" rstar="real[1]"
  amplitude="real[1]" href="str">
```

Define equaiton of state surface boundaries. Input is similar to the `<Path>` specification, but leg termination time is not specified. Control parameters also differ, as shown in Table 4.2.

cfmt	Variable type
1	Density
2	Temperature

Table 4.2: Supported surface variable types and `cfmt` code

## Examples

The following examples demonstrate the `<Surface>` input.

### format: default

```
<Surface>
  <!-- nsteps, control, Cij -->
  <!-- control goes as 1 -> density
                        2 -> temperature -->

    0 12 1 100
    100 12 5 300
</Surface>
```

### format: table

```
<Surface format="table" cfmt="12" nfac="100">
  <!-- Cij -->
    1 100
    5 300
</Surface>
```

## 4.6.3 Material

```
<Material model="str">
```

Specify the material model and parameters. Subelements of `<Material>` are

- `<Matlabel>`
- `<Key>`

Where `<Key>` is a valid material parameter name.

## Matlabel

```
<Matlabel href="str[F_MTL_PARAM_DB]">
```

Insert model parameters from a database file. The default file F\_MTL\_PARAM\_DB is in /path/to/gmd/materials/material\_properties.db.

## Key

```
<Key> float </Key>
```

## Examples

```
<Material model="elastic">
```

```
  <G> 54E+09 </G>
```

```
  <K> 124E+09 </K>
```

```
</Material>
```

```
<Material model="elastic">
```

```
  <Matlabel href="./materials.xml"> aluminum </Matlabel>
```

```
  <K> 124E+09 </K>
```

```
</Material>
```

### 4.6.4 Extract

```
<Extract format="str[ascii]{ascii, mathematica}" step="int[1]" fmt="str[.18f]">
```

Extract variables and paths from exodus output and (optionally) write to different formats. Recognized subelements of <Extract> are

- <Path>
- <Variables>

## Variables

```
<Variables> VAR_1, ..., VAR_N </Variables>
```

Variables to extract from the exodus output database. Variables are specified children of the <Variables> element. All components of vector and tensor variables will be extracted if only the basename is specified. Time is always extracted as the first entry of the output file. Extracted variables are in runid.out or runid.math depending if the format is ascii or mathematica.

## Path

### Supporting Drivers: eos

```
<Path type="str{isotherm, hugoniot}" increments="int[100]"
      density_range="list" initial_temperature="real">
```

Extract a specified path from the equation of state surface through the specified density range starting at the initial temperature.

### Examples

Extract all components of stress and strain

```
<Extract format="ascii">
  <variables>
    STRESS STRAIN
  </variables>
</Extract>
```

Extract only the XX, YY, and ZZ components of stress

```
<Extract format="ascii">
  <variables>
    STRESS_XX STRESS_YY STRESS_ZZ
  </variables>
</Extract>
```

Extract all variables

```
<Extract format="ascii">
  <variables>
    ALL
  </variables>
</Extract>
```

Extract Hugoniot and Isotherm paths

```
<Extract>
  <Path type="isotherm" increments="200"
        density_range="1 3" initial_temperature="225"/>
  <Path type="hugoniot" increments="100"
        density_range="1 3" initial_temperature="100"/>
</Extract>
```

## 4.7 Permutation

```
<Permutation method="str[zip]{zip, combine}" seed="real[12]">
```

Permutate model input parameters to investigate sensitivities. Recognized subelements of <Permutation>

- <Permutate>

### 4.7.1 Permutate

```
<Permutate var="str"  
          values="str{range, list, weibull, uniform, normal, percentage}"
```

Specify the parameters to permute. Variable names should occur elsewhere in the input file in preprocessing braces.

### 4.7.2 Example

Permutate the K and G parameters

```
<Permutation method="zip" seed="12">  
  <Permutate var="K" values="weibull(125.e9, 14, 3)"/>  
  <Permutate var="G" values="percentage(45.e9, 10, 3)"/>  
</Permutation>
```

In the <Material> element, the K and G parameters are specified as

```
<Material model="elastic">  
  <K> {K} </K>  
  <G> {G} </G>  
</Material>
```

## 4.8 Optimization

```
<Optimization method="str[simplex]{simplex, powell, cobyla}"  
              maxiter="int[25]" tolerance="real[1e-6]">
```

Optimize specified parameters against user specified objective function. Recognized subelements of <Optimization>

- <Optimize>
- <AuxiliaryFile>
- <ObjectiveFunction>

### 4.8.1 Optimize

```
<Optimize var="str" initial_value="real" bounds="list[]" />
```

Specify the variable to be optimized, giving initial value and, optionally, bounds. Only the `cobyla` method accepts bounds. Variable names should occur elsewhere in the input file in preprocessing braces.

### 4.8.2 AuxiliaryFile

Path to any auxiliary file needed by the optimization objective function.

```
<AuxiliaryFile href="str"/>
```

### 4.8.3 ObjectiveFunction

Path to a user defined executable script that returns the error to the optimization routine.

```
<ObjectiveFunction href="str"/>
```

An `ObjectiveFunction` given by `href` is called from the command line as

```
% ./scriptname simulation_output.exo [auxiliary_file_1 [... auxiliary_file_n]]
```

### 4.8.4 Example

Optimize the `K` and `G` parameters

```
<Optimization method="simplex" maxiter="25" tolerance="1e-4" disp="0">
  <ObjectiveFunction href="opt-sig-v-time"/>
  <AuxiliaryFile href="opt-baseline.dat"/>
  <Optimize var="opt_k" initial_value="129.e9"/>
  <Optimize var="opt_g" initial_value="54.e9"/>
</Optimization>
```

In the `<Material>` element, the `K` and `G` parameters are specified as

```
<Material model="elastic">
  <K> {opt_k} </K>
  <G> {opt_g} </G>
</Material>
```