

PARMEC MANUAL

October 18, 2016

Contents

| | | |
|----------|-----------------------|-----------|
| 1 | Running | 2 |
| 2 | Input commands | 3 |
| 2.1 | RESET | 3 |
| 2.2 | MATERIAL | 3 |
| 2.3 | SPHERE | 3 |
| 2.4 | MESH | 4 |
| 2.5 | ANALYTICAL | 4 |
| 2.6 | OBSTACLE | 5 |
| 2.7 | SPRING | 5 |
| 2.8 | GRANULAR | 6 |
| 2.9 | CONSTRAIN | 6 |
| 2.10 | PRESCRIBE | 7 |
| 2.11 | VELOCITY | 7 |
| 2.12 | GRAVITY | 7 |
| 2.13 | CRITICAL | 7 |
| 2.14 | DEM | 8 |
| 2.15 | HISTORY | 8 |
| 2.16 | OUTPUT | 8 |
| 3 | Output files | 10 |
| 4 | Output viewers | 11 |
| 5 | Tutorial | 12 |

Chapter 1

Running

PARMEC is a command line program. Typical usage:

1. Place PARMEC in a globally accessible path (e.g. suitably extend the PATH variable on a unix system).
2. Create a directory where your input file and output files will be stored (e.g. *mkdir test*).
3. Edit your Python input file in this directory (e.g. *test.py*); Chapter 2 documents all input commands.
4. Run PARMEC (e.g. *PARMEC path/to/test/test.py*).
5. Time histories can be generated during analysis using the HISTORY command; see Section 2.15.
6. Upon termination a *.dump file and/or a *.vtk.* file(s) is/are created in the same directory (e.g. *path/to/test/test.dump*) their format is documented in Chapter 3.
7. The output files can be viewed with OVITO and/or ParaView, as documented in Chapter 4.

A tutorial is provided in Chapter 5.

Chapter 2

Input commands

PARMEC input language extends Python. Subroutines related to input processing are listed below.

2.1 RESET

Erase all data.

RESET ()

2.2 MATERIAL

Create material.

matnum = MATERIAL (density, young, poisson)

- **matnum** - material number
- **density** - mass density
- **young** - Young modulus
- **poisson** - Poisson ratio

2.3 SPHERE

Create a spherical particle.

parnum = SPHERE (center, radius, material, color)

- **parnum** - particle number
- **center** - tuple (x, y, z) defining the center
- **radius** - radius
- **material** - material number
- **color** - positive integer surface color

2.4 MESH

Create a meshed particle.

parnum = MESH (nodes, elements, material, colors)

- **parnum** - particle number
- **nodes** - list of nodes: $[x0, y0, z0, x1, y1, z1, \dots]$
- **elements** - list of elements: $[e1, n1, n2, \dots, ne1, me1, e2, n1, n2, \dots, ne2, me2, \dots]$, where $e1$ is the number of nodes of the first element, $n1, n2, \dots, ne1$ enumerate the element nodes, and $me1$ is the material number. Similarly for the second and all remaining elements. Supported numbers of nodes per element are 4, 5, 6, and 8 for respectively *tetrahedron*, *pyramid*, *wedge*, and *hexahedron*, cf. Figure 2.1.
- **material** - material number
- **colors** - list of positive integer face colors: $[gcolor, f1, n1, n2, \dots, nf1, c1, f2, n1, n2, \dots, nf2, c2, \dots]$, where $gcolor$ is the global color for all not specified faces, $f1$ is the number of nodes in the first specified face, $n1, n2, \dots, nf1$ enumerate the face nodes, and $c1$ is the surface color of that face. Similarly for the second and all remaining faces. If only the global color is required, it can be passed as $[gcolor]$ or as $gcolor$ alone.

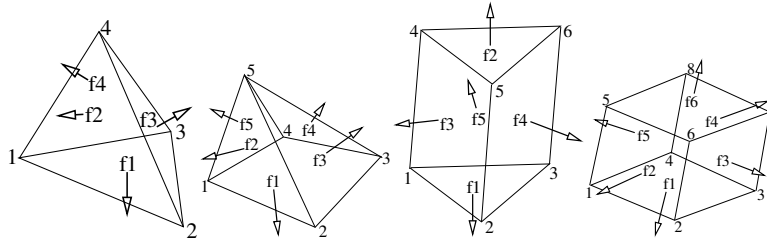


Figure 2.1: Element types.

2.5 ANALYTICAL

Create an analytical particle. Analytical particles have no shapes and are not involved in contact.

parnum = ANALYTICAL (| inertia, mass, rotation, position, material, particle)

Note, that all parameters are optional.

- **parnum** - particle number
- **inertia** - inertia tensor passed as a list $[Ixx, Iyy, Izz, Ixy, Ixz, Iyz]$; optional, if **particle** parameter is used; default $[1, 1, 1, 0, 0, 0]$
- **mass** - scalar mass; optional, if **particle** parameter is used; default 1
- **rotation** - optional orientation matrix passed as a list $[e1x, e1y, e1z, e2x, e2y, e2z, e3x, e3y, e3z]$, where vectors $e1, e2, e3$ are orthonormal; default $[1, 0, 0, 0, 1, 0, 0, 0, 1]$
- **position** - optional position vector passed as a tuple (x, y, z) ; default $(0, 0, 0)$

- **material** - material number; default 0
- **particle** - optional; if specified, an existing particle is converted into an analytical particle; its properties are inherited or overwritten, depending on whether any of the **inertia**, **mass**, **rotation**, **position** parameters are used; if initially specified, particle shape is inherited and its animated motion is included into the results

2.6 OBSTACLE

Create an obstacle.

OBSTACLE (triangles, color | point, linear, angular)

- **triangles** - list of triangle tuples $[(t1x1, t1y1, t1z1, t1x2, t1y2, t1z2, t1x3, t1y3, t1z3), (t2x1, t2y1, t2z1, t2x2, t2y2, t2z2, t2x3, t2y3, t2z3), \dots]$ defining the obstacle
- **color** - positive integer surface color or a list $[color1, color2, \dots]$ of colors for each individual triangle
- **point** - spatial reference point
- **linear** - linear velocity history callback: $(v_x, v_y, v_z) = \mathbf{linear}(t)$
- **angular** - spatial angular velocity history callback: $(\omega_x, \omega_y, \omega_z) = \mathbf{angular}(t)$

2.7 SPRING

Create a translational spring constraint. The applied force formula reads

$$\text{force}(t) = \text{direction}(t) \cdot [\text{spring}(\text{stroke}(t)) + \text{dashpot}(\text{velocity}(t)) \cdot |\text{sign}(\text{spring}(\text{stroke}(t)))|]$$

where

$$\text{direction}(t) = (\text{point2}(t) - \text{point1}(t)) / |\text{point2}(t) - \text{point1}(t)| \text{ or constant } (d_x, d_y, d_z) \text{ or tangent}$$

$$\text{stroke}(t) = \text{direction}(t) \cdot [\text{point2}(t) - \text{point1}(t)] - \text{direction}(0) \cdot [\text{point2}(0) - \text{point1}(0)]$$

$$\text{velocity}(t) = \text{direction}(t) \cdot \frac{d}{dt} [\text{point2}(t) - \text{point1}(t)]$$

$$\text{sign}(x) = \begin{cases} -1 & \text{if } x < 0 \\ 0 & \text{if } x = 0 \\ 1 & \text{if } x > 0 \end{cases}$$

The $\text{spring}(\text{stroke})$ and $\text{dashpot}(\text{velocity})$ relationships are defined by means of lookup tables; $\text{force}(t)$ is applied at $\text{point2}(t)$, and $-\text{force}(t)$ is applied at $\text{point1}(t)$; dashpot force is not applied when spring force is zero.

sprnum = **SPRING** (**part1**, **point1**, **part2**, **point2**, **spring** | **dashpot**, **direction**, **planar**)

- **sprnum** - spring number
- **part1** - first particle number
- **point1** - tuple (x, y, z) defining a point moving with the first particle
- **part2** - second particle number; -1 can be used to indicate a single-particle constraint
- **point2** - tuple (x, y, z) defining a second point, either moving with the second particle, or a spatial point
- **spring** - spring force lookup table $[\text{stroke}_1, \text{force}_1, \text{stroke}_2, \text{force}_2, \dots, \text{stroke}_n, \text{force}_n]$
- **dashpot** - optional dashpot force lookup table $[\text{velocity}_1, \text{force}_1, \text{velocity}_2, \text{force}_2, \dots, \text{velocity}_m, \text{force}_m]$; default: $[-\infty, 0, +\infty, 0]$
- **direction** - optional constant direction (d_x, d_y, d_z)
- **planar** - optional planar spring flag; when 'ON' spring direction

$$(\text{point2}(t) - \text{point1}(t)) / |\text{point2}(t) - \text{point1}(t)|$$

is projected onto a plane orthogonal to (d_x, d_y, d_z) ; default: 'OFF'

2.8 GRANULAR

Define surface pairing for the granular contact interaction model.

GRANULAR (**color1**, **color2**, **spring** | **damper**, **friction**, **rolling**, **drilling**, **kskn**)

- **color1** - first color (positive, or $\text{color1} = 0$ and $\text{color2} = 0$ to redefine default parameters)
- **color2** - second color (positive, or $\text{color1} = 0$ and $\text{color2} = 0$ to redefine default parameters)
- **spring** - normal spring constant
- **damper** - optional normal damping ratio; default: 1.0
- **friction** - optional Coulomb's friction coefficient; default: 0.0; tuple (μ_s, μ_d) can be used to specify respectively static and dynamic friction coefficients
- **rolling** - optional rolling friction coefficient; default: 0.0
- **drilling** - optional drilling friction coefficient; default: 0.0
- **kskn** - optional ratio of normal to tangential spring and dashpot parameters; default: 0.5

2.9 CONSTRAIN

Constrain particle motion.

CONSTRAIN (parnum | linear, angular)

- **parnum** - particle number
- **linear** - list $[x_1, y_1, z_1]$, $[x_1, y_1, z_1, x_2, y_2, z_2]$, or $[x_1, y_1, z_1, x_2, y_2, z_2, x_3, y_3, z_3]$ defining directions of constrained linear motion; default: $[0, 0, 0]$
- **angular** - list $[x_1, y_1, z_1]$, $[x_1, y_1, z_1, x_2, y_2, z_2]$, or $[x_1, y_1, z_1, x_2, y_2, z_2, x_3, y_3, z_3]$ defining directions of constrained spatial rotation; default: $[0, 0, 0]$

2.10 PRESCRIBE

Prescribe particle motion. Prescribed motion overwrites this resulting from dynamics and constraints.

PRESCRIBE (parnum | linear, angular, kind)

- **parnum** - particle number
- **linear** - linear velocity or acceleration history callback: $(v_x, v_y, v_z) = \mathbf{linear}(t)$; default: *not prescribed*
- **angular** - spatial angular velocity or acceleration history callback: $(\omega_x, \omega_y, \omega_z) = \mathbf{angular}(t)$; default: *not prescribed*
- **kind** - string 'vv', 'va', 'av', or 'aa' indicating interpretation of respectively **linear** and **angular** time histories as either velocity or acceleration; default: 'vv'

2.11 VELOCITY

Set particle velocity.

VELOCITY (parnum | linear, angular)

- **parnum** - particle number
- **linear** - linear velocity tuple (v_x, v_y, v_z) ; default: $(0, 0, 0)$ at $t = 0$
- **angular** - angular velocity tuple $(\omega_x, \omega_y, \omega_z)$; default: $(0, 0, 0)$ at $t = 0$

2.12 GRAVITY

Set gravity.

GRAVITY (gx, gy, gz)

- **gx** - constant x component
- **gy** - constant y component
- **gz** - constant z component

2.13 CRITICAL

Estimate critical time step.

h = CRITICAL ()

- **h** - critical time step

2.14 DEM

Run DEM simulation.

t = DEM (duration, step | interval, prefix)

- **t** - simulation runtime in seconds
- **duration** - simulation duration
- **step** - time step
- **interval** - output interval (default: time step); tuple $(dt_{\text{files}}, dt_{\text{history}})$ can be used to indicate different output frequencies of output files and time histories
- **prefix** - output file name prefix (default: input file name without the “.py” extension)

2.15 HISTORY

Request time history output.

list = HISTORY (entity | source, point)

- **list** - output time history list
- **entity** - entity name; one of: (output time) 'TIME', (position) 'PX', 'PY', 'PZ', '|P|', (displacement) 'DX', 'DY', 'DZ', '|D|', (linear velocity) 'VX', 'VY', 'VZ', '|V|', (angular velocity) 'OX', 'OY', 'OZ', '|O|', (body force) 'FX', 'FY', 'FZ', '|F|', (body torque) 'TX', 'TY', 'TZ', '|T|'
- **source** - particle number i , or a list of particle numbers $[i, j, \dots]$, or a spatial sphere defined as tuple (x, y, z, r) , or a spatial box defined as tuple $(x_{\min}, y_{\min}, z_{\min}, x_{\max}, y_{\max}, z_{\max})$; in case of a list of particle numbers the output entity is averaged over the set of particles; in case of a spatial sphere or box the output entity is averaged over the set of particles passing through it; default: 0 (useful when entity is 'TIME')
- **point** - optional referential point used in case of a single particle source; default: particle mass centre

2.16 OUTPUT

Define scalar and/or vector entities included into the output file(s). PARMEC outputs:

- *.dump files for spherical particles
- *0.vtk.* files for obstacles and mesh based particles **not** specified as **a subset** in the OUTPUT command
- *1.vtk.*, *2.vtk.*, ... files for mesh based particles specified as subsets, where numbers 1, 2, ... match consecutive OUTPUT calls
- *0rb.vtk.* for rigid body data of particles **not** specified as **a subset** in the OUTPUT command

- *1rb.vtk.*, *2rb.vtk.*, ... files for rigid body data of particles specified as subsets, where numbers 1, 2, ... match consecutive OUTPUT calls
- *0cd.vtk.* for contact data including particles **not** specified as a **subset** in the OUTPUT command
- *1cd.vtk.*, *2cd.vtk.*, ... files for contact data including particles specified as subsets, where numbers 1, 2, ... match consecutive OUTPUT calls
- *0sd.vtk.* for spring data including particles **not** specified as a **subset** in the OUTPUT command
- *1sd.vtk.*, *2sd.vtk.*, ... files for spring data including particles specified as subsets, where numbers 1, 2, ... match consecutive OUTPUT calls

OUTPUT (entities | subset, mode)

- **entities** - list of output entities; default: ['NUMBER', 'COLOR', 'DISPL', 'ORIENT', 'LINVEL', 'ANGVEL', 'FORCE', 'TORQUE', 'F', 'FN', 'FT', 'SF', 'AREA', 'PAIR'] where:
 - 'NUMBER' - scalar field of particle numbers (modes: 'SPH', 'MESH', 'RB'), or scalar field of spring numbers (modes: 'SD')
 - 'COLOR' - scalar field of surface colors (modes: 'SPH', 'MESH'), or 2-component vector field of contact surface colors (modes: 'CD')
 - 'DISPL' - 3-component vector field of displacements (modes: 'SPH', 'MESH', 'RB'), or scalar field of contact depths (modes: 'CD'), or scalar field of spring strokes (modes: 'SD')
 - 'ORIENT' - three 3-component vector fields representing columns of rigid rotation matrix (orientation vectors) (modes: 'RB'), or 3-component vector field of spring orientations (modes: 'SD')
 - 'LINVEL' - 3-component vector field of linear velocity (modes: 'SPH', 'MESH', 'RB')
 - 'ANGVEL' - 3-component vector field of (spatially constant) angular velocity (modes: 'SPH', 'MESH', 'RB')
 - 'FORCE' - 3-component vector field of (spatially constant) total body force (modes: 'SPH', 'MESH', 'RB')
 - 'TORQUE' - 3-component vector field of (spatially constant) total body torque (modes: 'SPH', 'MESH', 'RB')
 - 'F' - 3-component vector field of total contact forces (modes: 'CD'), or scalar field of total spring forces (modes: 'SD')
 - 'FN' - 3-component vector field of normal contact forces (modes: 'CD')
 - 'FT' - 3-component vector field of tangential contact forces (modes: 'CD')
 - 'SF' - scalar field of spring force magnitude, without dashpot contribution (modes: 'CD', 'SD')
 - 'AREA' - scalar field of contact area (modes: 'CD')
 - 'PAIR' - 2-component vector field of particle pair numbers (modes: 'CD', 'SD')
- **subset** - optional particle number i , or a list of particle numbers $[i, j, \dots]$, to which this specification is narrowed down
- **mode** - optional output mode or list of output modes: 'SPH' for sphere output, 'MESH' for mesh output, 'RB' for rigid body output, 'CD' for contact data output, 'SD' for spring data output; default: ['SPH', 'MESH', 'RB', 'CD', 'SD']

Chapter 3

Output files

Chapter 4

Output viewers

Chapter 5

Tutorial