

## BETA Documentation (*Elasticity Analysis*)

To complete a periodic textile composite elasticity analysis the following files are needed:

- Beta input file (.beta)
- Mesh File (.plt)
- Material Library File (.matlib)
- Element Material File (.elemat)
- Material Angles File (.mangles)

All or some of the following files can be output by BETA:

- Beta output file (output.txt)
- Time log file (time log.txt)
- Nodal Displacement File (disp)
- Nodal Stress/Strain Files
- Quadrature Point Stress Strain Files
- Volume Average Stress File (volumeAverage)

### **Input Files:**

#### **Beta Input File (.beta)**

```
createModel          //This block creates the model used for analysis.
  ElasticityModel    //In this case an elasticity model.
exitcreateModel

ReadTitle            //This block assigns a title to the analysis.
pw_uc.beta

setVerboseFlag      0          //This value controls the amount of intermediate info output by beta.
SolverSettings
  setSolverVerboseFlag 1        //This value controls the amount of info output during solving.
  setStorageMethod mklpardiso //This specifies the solver to be used in the analysis.
  setSparseSolverMaxIterations 100000 //For iterative solvers this specifies the max number of iterations.
  setSparseSolverTolerance 1e-8  //For iterative solvers this specifies the tolerance.
  replaceZeroDiagonal 1.0        //Specifies value to replace zeroes on the diagonal of K matrix.
  UseMultiCore          max      //Specifies number of cores to use for solver(all cores in this case).
exitSolverSettings

UseMultiCore          max      //Specifies number of cores to use outside solver (all cores in this
                                case)
ParallelAssembly      0        //Turns on (1) or off (0) Parallel Assembly. This option is still under
                                development. To ensure correct results turn parallel assembly off (0).

BasicElement::SetAnalysisType //This option is for analysis using geometric non-linearity.
0                             //A value of 0 turns non-linearity off.

maxResidual           //Again, this option is for geometric non-linearity
0.01

SetFindNodeTOLERANCE 1e-8      //Sets the tolerance for operations that pick nodes by coordinate.
SetMPCTransformationTolerance 1e-5 //Sets the tolerance for MPC Transformations
```



```
CreateElements                                     //This block creates the elements used in analysis.
getNumberOfElementsFromMesh pw_uc.plt             //Retrieves number of elements from mesh file.
ElasticityElement3D                               //Element type.
all 1                                              //sets all elements to specified element type
-1
exitCreateElements

openFile carbon_epoxy.matlib ReadMaterials         //Opens and reads material properties from material library
                                                    file.

openFile pw_uc.plt ReadMesh                       //Opens and reads mesh file.

setNumDofPerNode                                  //Sets the number of degrees of freedom(DOF) per node
3 all                                              //Sets all nodes to have 3 DOF
6 42 42 1                                         //# DOF, first node, last node, increment (in this case node #42
                                                    is a dummy node used for periodic boundary conditions)
-1

ReadMultiPointConstraints

PlaneToPlane
Coord 0.0 -7.5e-001 5.0e-001 0.0 -7.5e-001 0.0 0.0 0.0 0.0
Coord 0.0 -7.5e-001 -5.0e-001 0.0 -7.5e-001 0.0 0.0 0.0 0.0
//specify equations between master and slave
0 0 -1 0.0 // slave DOF = master DOF * -1 + 0.0
1 1 1 0.0
2 2 1 2 1.0 0.0 // slave DOF = (master DOF * 1) + (dummy DOF * 1.0) + 0.0
exit

//Multipoint constraints (MPCs) may be specified between "plane to plane", "plane to line",
//"plane to point", "line to point", and "point to point". The first coordinate set specifies coordinates
//for the slave plane, line, or point. The second coordinate set specifies coordinates for the master
plane,
//line, or point. Of course, multiple MPCs may be specified within this block.

exitReadMultiPointConstraints
```



```
//Boundary Conditions
CreateNodeGroup          //This command creates groups of nodes to be used in constraints/loading.
  origin Point 0.0 0.0 0.0 //Node group name, node group type, coordinate(s).
exitCreateNodeGroup

readConstraints
  origin 1 2 3          //Constrains the first, second, and third DOF of nodes in the node group
                        "origin".
exitreadConstraints

openFile pw_uc.eleamat SetElementProperty //Opens the element material file and reads material
                                         assignments.

SetElementProperty
SetIntegrationOrder      //This command sets the integration order used for elements.
all 3
-1 0 0

SetElementProperty
SetElementTemperature    //This command sets the element temperature (not used here).
-1 0 0

SetElementProperty
SetElementMoisture       //This command sets the element temperature (not used here).

-1 0 0

openFile pw_uc.mangles readSpecialElementCommand //Opens and reads the material angles file.

ReadLoads
DisplacementLoad         //Assigns a displacement load to the first DOF of node # 42.
42 0.01 1               //In this case we are actually specifying a volume average displacement gradient
-1                      //using the "dummy node".
exitReadLoads
```



```
ReadOptionalOutput          //Optional post processing output
  volumeAverage             //volume averaged stress and strain
  displacements             //nodal displacements
  LCS_stress                 //local coordinate system (material coordinate system) stress
  LCS_strain                 //local coordinate system (material coordinate system) strain
  GCS_stress                 //global coordinate system stress
  GCS_strain                 //global coordinate system strain
  Quad_stress                //quadrature point stress
  Quad_strain                //quadrature point strain
exitReadOptionalOutput

DoAnalysis 0                //Begins the finite element analysis.

end                          // end of input file
```

### Mesh File (.plt)

```
43  4  3  // # of Nodes    # of Elements    # of Dimensions
0    0.0 -7.5e-001  5.0e-001 //Node #      x    y    z
1    0.0 -7.5e-001  3.75e-001
.
.
.
41  -7.5e-001  0.  -3.75e-001
42  0.1  0.1  0.1
0 20 2 18 19 20 21 22 4 3 7 23 24 8 12 12 12 13 14 14 14 13
//Element #, # Nodes per element, Connectivity
1 20 19 19 19 25 26 27 21 20 23 23 28 24 12 12 12 29 30 31 14 13
2 20 0 1 2 3 4 4 4 5 6 7 8 9 10 11 12 13 14 15 16 17
3 20 19 32 33 34 26 26 26 25 23 35 36 28 12 37 38 39 40 41 30 29
```



### Material Library File (.matlib)

```
ElasticMaterial          //Material type.
1 carbon+epoxy warp      //Material number and description.
readModuli
157.95e9 9.027e9 9.027e9 //E11 E22 E33
0.2412 0.3749 0.2412    //nu12 nu23 nu13
5.12e9 3.34e9 5.12e9    //G12 G23 G13
exitElasticMaterial

ElasticMaterial
2 carbon+epoxy fill
readModuli
157.95e9 9.027e9 9.027e9
0.2412 0.3749 0.2412
5.12e9 3.34e9 5.12e9
readAngles               //This command rotates the material properties.
1 3 90                  //# of angle rotations  rotation axis  rotation degrees
exitElasticMaterial

ElasticMaterial
3 Neat Epoxy 411-350
readModuli
3.1e9 3.1e9 3.1e9
0.35 0.35 0.35
1.15e9 1.15e9 1.15e9
exitElasticMaterial

exitReadMaterials
```

### Element Material File (.elemat)

The ELEMAT file associates material groups with elements.

```
selectElementMaterial
0 0 1 1          //starting element #    ending element #    increment    material #
1 1 1 2
2 3 1 3
-1 0 0
```

### Material Angles File (.mangles)

The MANGLES file specifies the material orientation for an undulating tow. An axis of rotation and an angle is specified for each node in an element.

```
setElementMaterialAngles
0 2      27.6365 27.6365 27.6365 20.3166 2.05414e-011 2.05414e-011 2.05414e-011 20.3166 27.6365 27.6365
        2.05414e-011 2.05414e-011 27.6365 27.6365 27.6365 20.3166 2.05414e-011 2.05414e-011 2.05414e-011
        20.3166
// Element #    rotation axis    material orientation at node
1 1      2.05414e-011 2.05414e-011 2.05414e-011 2.05414e-011 2.05414e-011 2.05414e-011 2.05414e-011 2.05414e-
011
        20.3166 20.3166 20.3166 20.3166 27.6365 27.6365 27.6365 27.6365 27.6365 27.6365 27.6365 27.6365
2 3      0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
3 3      0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
```

**Output Files:**

**Beta Output File (Output.txt):**

*This file contains information pertaining the execution of the finite element analysis.  
It is useful for troubleshooting in the event that an analysis does not run as expected.*

**Time Log File (timelog.txt):**

*This file shows the time required for particular processes of the finite element analysis.*

**Nodal Displacement File (disp):**

```
displacements
// node number      x displacement      y displacement      z displacement
0                   -6.8911303789130742e-003  9.2503040095788188e-004  -3.1232930736917160e-003
1                   -6.8853971455278638e-003  9.2503040095788188e-004  -3.2895864261015444e-003
2                   -6.8795271396461504e-003  9.2503040095788188e-004  -3.4557682465640072e-003
....
```



## Nodal Stress/Strain File:

stress

```
0 1 //element number, material group number
// xx component yy comp zz comp xy comp yz comp xz comp node number
7.4709e+008 5.5387e+007 2.1535e+008 -3.4356e+005 -2.3650e+006 -6.3463e+007 19
6.9641e+008 1.5897e+007 1.3103e+008 7.1608e+005 -2.5087e+005 -2.4006e+007 28
8.2835e+008 -6.2307e+006 5.6946e+007 -4.8014e+005 -1.8377e+006 1.3031e+007 29
8.3872e+008 4.4878e+005 8.0371e+007 -1.3914e+005 -2.1807e+006 4.8391e+006 30
7.6031e+008 -2.7919e+006 8.0024e+007 1.9305e+003 -2.5786e+006 -3.3187e+006 31
6.7568e+008 3.3953e+007 1.8041e+008 -5.8312e+004 6.6599e+005 1.8747e+005 32
7.5889e+008 9.6628e+007 2.9620e+008 -1.1511e+005 3.7932e+004 -1.8422e+006 4
7.1353e+008 8.1408e+007 2.7033e+008 -7.3679e+005 -2.6555e+006 -3.4472e+007 20
7.2294e+008 5.0116e+007 2.0378e+008 1.6995e+006 -3.1594e+006 -5.8902e+007 21
8.2194e+008 -1.0926e+006 6.1187e+007 4.0294e+006 -2.8274e+006 1.5438e+007 33
7.6000e+008 4.9949e+006 8.6213e+007 -7.8239e+004 -5.1133e+006 -2.5742e+006 34
7.0874e+008 8.6214e+007 2.8055e+008 -1.6130e+006 -3.2042e+006 -6.4410e+006 22
6.6290e+008 4.0497e+007 1.8465e+008 4.5999e+006 -5.4713e+006 -4.8802e+007 24
6.9390e+008 9.1540e+006 1.1281e+008 9.9968e+006 -5.6084e+006 -1.3735e+007 35
8.2583e+008 -3.0342e+006 5.0971e+007 7.9675e+006 -4.9357e+006 1.8373e+007 36
8.3987e+008 2.7571e+006 6.9340e+007 3.8030e+006 -7.3706e+006 8.2861e+006 37
7.6814e+008 2.0269e+006 7.0647e+007 -7.3989e+005 -9.6049e+006 -1.7944e+006 38
6.6416e+008 2.0519e+007 1.5225e+008 9.5666e+005 -1.0208e+007 -9.3663e+005 39
6.4808e+008 6.9415e+007 2.5033e+008 -1.8619e+006 -8.9091e+006 -4.7007e+006 26
6.3895e+008 5.8899e+007 2.2837e+008 2.0150e+006 -8.1181e+006 -2.8356e+007 25
...
```

### Quadrature Point Stress Strain File (Quad.stress/strain):

For a 3D analysis this file will contain 17 columns. The first 6 columns are the local stress/strain components at a particular quadrature point (xx, yy, zz, xy, yz, xz). The next 6 columns are the global stress/strain components at a particular quadrature point. The next column contains the volume associated with that quadrature point. The next column contains the material group associated with the quadrature point. The last three columns contain the x, y, and z coordinates of the quadrature point respectively.

To summarize:

**L\_xx L\_yy L\_zz L\_xy L\_yx L\_xz G\_xx G\_yy G\_zz G\_xy G\_yx G\_xz Vol Mat# x y z**

### Volume Averaged Stress/Strain File(volumeAverage):

```
//material group # (0 for whole model) , associated volume, -1, 0, 0, total Strain Energy Density
//<ε xx>, <ε yy>, <ε zz>, <ε xy>, <ε yz>, <ε xz>
//<σ xx>, <σ yy>, <σ zz>, <σ xy>, <σ yz>, <σ xz>
```

```
0      0.5625      -1      0      0      907019
0.01    -0.00123337    -0.00624659    0.000177975    0.00149698    0.000903577
3.22496e+008  -1.28786e-006  -1.1911e-005    -260068    -57844    -7.5223e+007
```

...