```
/*    *.fem  the data file, contain mesh information, for FEM:

      F_X.n, F_X.d, F_T.n
      x, y, [z,] (i)      (F_X.x: x[n],y[n],z[n])
       :                (i=0, F_X.n-1)
      n, element type, (i)
      materal type, i1,i2,i3,[i4,[i5,i6,[i7,i8,]]] (j) (j=0, n-1)
       :                (i=0, F_T.n-1)
      (end)

      F_X.n,F_X.d,F_T.n: The number of node, dimension, and type of element
      n: number of element for each type
      element type: plane ele(2001,2002,2003,2004), bar ele(2101,2102), beam
                ele(2201,2202), plane contact ele with slid property in
                0--1 direction(2501,2502,2503,2504), bar ele with slid
                property(2601,2602)
      material type: 1, 2, 3,... Start from one

      This file will be created by mesh generator.

      local coord num:  2001(2501)  2002(2502)  2003(2503)  2004(2504)
                              2             3   2             2             3 6 2
                                                            5 4            7   5
                            0     1           0   1          0 3 1          0 4 1

          Bar ele 2101:   2102:     Beam ele 2201:  2202:
                    0---1   0-2-1                 0---1   0-2-1
          Bar ele with slide property 2601:   2602:
                                        2---3    3-5-4  steel bar
                                        0    1     0 2 1  concrete node

*/


/*    *.mat  the data file, contain material property, for FEM:

      n, ipl      // (ipl: 0,plane stress; 1,plane strain; 3,3D)
      id, material property, (i)  //(i=0, n-1)
        :
        :
      (end)

      id: material property type.
        id=1: isotropic material. E, V, T, px, py,alpha
        id=2: orthotropic material. E11, v12/v21, G12, alpha1,
                                E22, v13/v31, G13, alpha2,
                                E33, v23/v32, G23, alpha3,
                                T, px, py
            v12/v21: v12, when E11 > E22     (v12/E11 = v21/E22)
```

```
                v21, when E11 < E22
            material coordinate system is same as the element local
            coordinate.
        id=3: for Bar element. E,A,I,px,py,pz,alpha
            where A is the area of section, I is minimum of I1 and I2.
        id=4; for Bar element with sliding properity. E,G,A,I,px,py,pz,alpha

    For example:
    4, 0            //(n=3, ipl=0)
    1, E, v, t, px, py, (1)
    2, E1, v12, g12, alpha1,
    E2, v13, g13, alpha2,
    E3, v23, g23, alph3,
    t, px, py, (2)
    3, E, A, I, px, py, pz, alpha, (3)
    4, Es, Gc, A, I, px, py, pz, alpha, (4)

    After read in:
    F_MAT= {
     n=4; ipl=0;
     Mats[0] = { id=1, n=5, e[6]=( E, v, t, px, py, alpha)};
     Mats[1] = { id=2, n=12, e[15]=( E11, v12/v21, G12,alpha1,
                          E22, v13/v31, G13,alpha2,
                          E33, v23/v32, G23,alpha3,
                          T, px, py )};
     Mats[2] = { id=3, n=5, e[7]=(E, A, I, px, py,pz,alpha)};
     Mats[3] = { id=4, n=6, e[8]=(Es,Gc,A,I,px,py,pz,alpha)};
    }
*/

/* file *.pqu:
    np0, np,nu
    k, P0k, (i)   (k: dof)
    :          (i=1,np0)
    k, Pk, (i)
    :          (i=1,np)
    k, Uk, (i)
    :          (i=1,nu)
    (end)

  np0,np,nu: number of dead load, live load and prescribed deformation in d.o.f
  k: degree of freedom
  P0k,Pk: dead load and live load in kth d.o.f
  Uk: prescribed deformation in kth d.o.f

*/
```