SLang - the Next Generation

Tutorial

Christian Bucher, Sebastian Wolff
Center of Mechanics and Structural Dynamics
Vienna University of Technology

November 8, 2010

0.1 Analysis of imported FE mesh

This example shows the import and analysis of a tetrahedral volume mesh generated by **gmsh**. The geometry is defined as shown in Fig. **??**. It is them meshed with 5515 4-node tetrahedral elements. The structure is

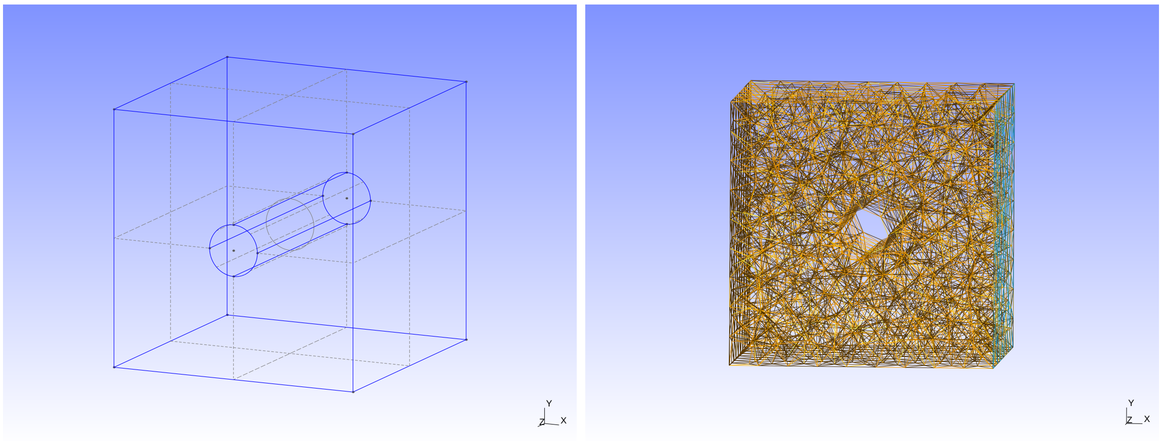

Figure 1: Geometry of block with cylindrical hole

supported on one side. The support elements are defined as physical group in **gmsh**. On the opposite side, a transverse load is applied (in y-direction).

The procedure to arrive at the solution of this problem is given in the following script.

```
--[[
SLangTNG
Test for Finite Element analysis
FE model imported from Gmsh
(c) 2009 Christian Bucher, CMSD-VUT
--]]


-- import the model (Tetrahedra vor volumes, triangles for surfaces) and set all DOF's to available
   struc=tngfem.TNGStructureImportGmsh("block.msh")
   struc:SetAvailDof(1, 1, 1, 1, 1, 1)

-- Get the element group containing the support surface and convert to node group
   support=struc:GetGroup(1)
   nsup=support:ToNodeGroup(101)

-- remove all availabled DOF's for support
   struc:SetAvailDof(0, 0, 0, 0, 0, 0, nsup:GetMemberList())

-- Get the element group carrying the distributed load (triangles)
   load=struc:GetGroup(2)
   loadList = load:GetMemberList()

-- Get the element group defining the body (tetrahedra)
   evol=struc:GetGroup(3)
   evolList = evol:GetMemberList()

-- Define section and material properties (Gmsh provides only the mesh)
   ss=struc:AddSection(301, "SHELL", 0, 0.01)
   ss:SetColor(0,200,200,255)
   struc:SetSection(301, loadList)
   struc:SetSection(301, support:GetMemberList())

   s=struc:AddSection(300, "VOLUME", 0)
   s:SetColor(255,0,0,255)
   struc:AddMaterial(800, "LINEAR_ELASTIC", 1, .3, 1)
   struc:SetMaterial(800, evolList)
   struc:SetSection(300, evolList)

-- Assign global DOF numbers
   nd=struc:GlobalDof()
```

```

-- define distributed load in global y-direction
  force=tmath.ReadMatrix({{0},{1},{0}})

-- Assemble global load vector
  F=struc:GlobalForce(force, loadList)

-- Assemble global stiffness matrix
  K=struc:SparseStiffness(evolList)

-- Solver for displacements
  U=K:Solve(F)

-- Show deformed structure (only volume elements are set visible)
  struc:SetDofDisplacements(U)

  vis=tnggraphics.TNGVisualize(40, 40, 1100, 800, "Structure")
  vis:Lighting(true)
  vis:Perspective(true)
  vis:SetAngles(20,-20,0)
  vis:Draw(struc, .05)

-- Add a vector plot showing the displacements
  U2 = struc:GetAllDisplacements()
  vis:Vector(struc, U2, .05)
  vis:File("block_def.pdf")
  vis:File("block_def.png")

--[[
  Compute and visualize stresses
  The stresses are computed in ElementStressresult(k...). Here
  the meaning of k is:
    0 v.Mises stress
    1 s_xx
    2 s_yy
    3 s_zz
    4 t_xy
    5 t_xz
    6 t_yz
--]]
  struc:SetVisible(false)
  struc:SetVisible(true, evolList)
  sv=tnggraphics.TNGSuperVisualize(40, 40, 1100, 800, "Stresses")
  for i=1,6 do
    v=sv:AddVisualize("Stress"..i, math.mod(i-1,2)==0)
    stress = struc:ElementStress(i)
    v:Perspective(true)
    v:Palette(true)
    v:Lighting(true)
    v:SetAngles(20,-10,0)
    v:ElementResult(struc, stress, true, 0.05)
    v:Zoom(1.3)
  end
  sv:File("block_stress.pdf", 3)
```

The deformed structure is shown in Fig. **??**. The stresses are shon in Fig. **??**.

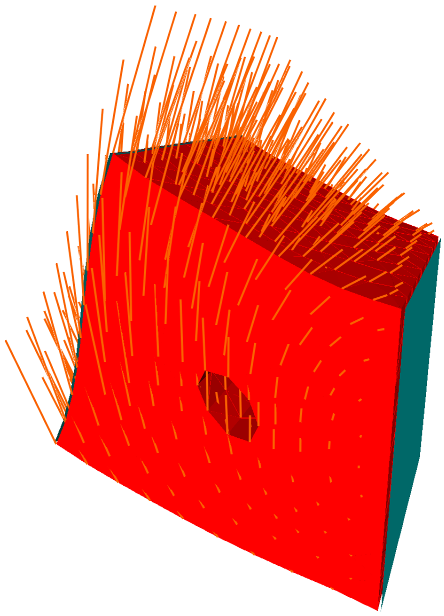

Figure 2: Deformation of block with cylindrical hole

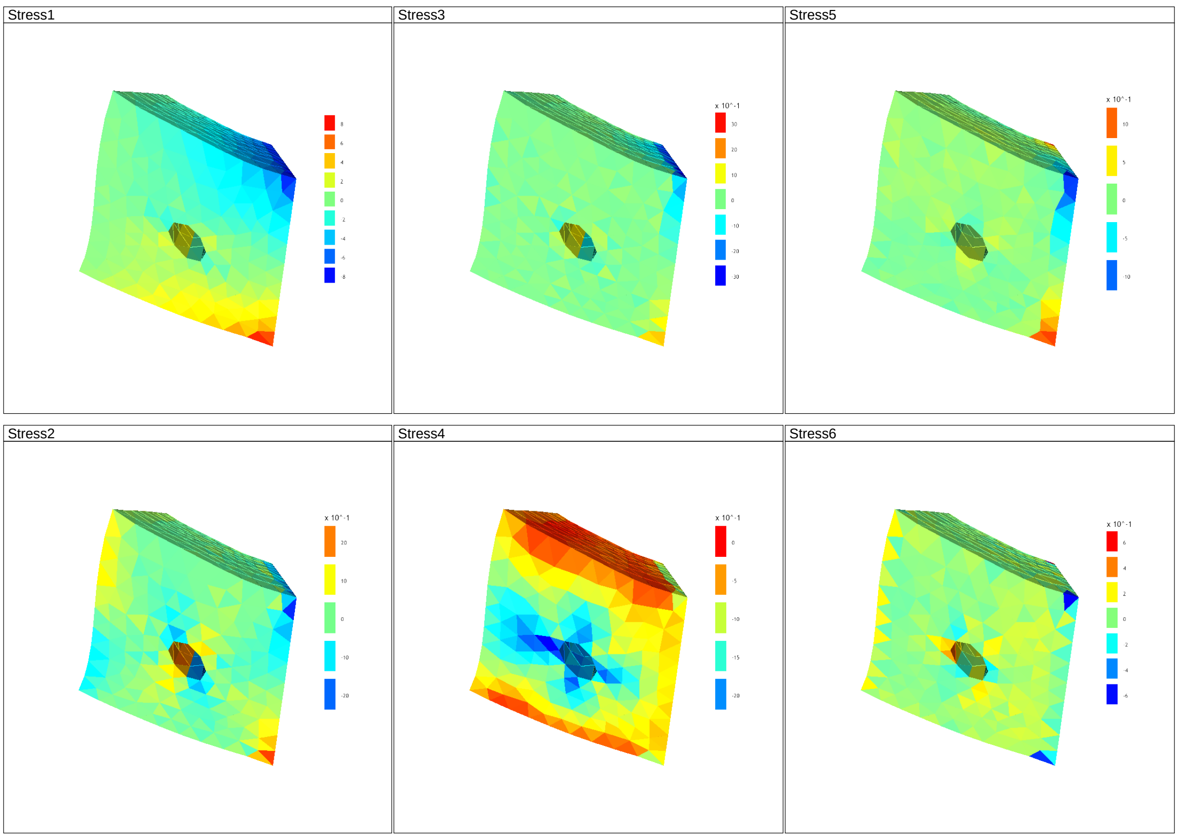

Figure 3: Stresses in block with cylindrical hole