SIMULATION OF A LUMBAR MOTION SEGMENT

Overview

This document shows a simulation of the creep deformation of an intervertebral disc (IVD). IVD is the smallest segment of the spine and is composed of soft tissues. Many experiments observations suggest considering the soft tissue as a biphasic model. Coupled pore-pressure-thermoelements can simulate the compressibility change due to fluid flow and thus are used to provide a realistic simulation. The creep behavior is taken care of by pore-pressure-thermoelements, which are defined through command snippet in Ansys Mechanical.

Goals

- To learn how to use coupled pore-pressure-thermoelements for IVD creep simulation

Steps

1. Import the mesh of the bones. Open Ansys workbench. Create an "External model" component. Import the mesh of the bone model.
2. Transfer the mesh information. Create a "Static Structural" system and link the “External model” to the model of “Static Structural”. Set the units to Metric (kg, mm, s, N). Define a material for the bone. The meshed model is shown in Figure 1.

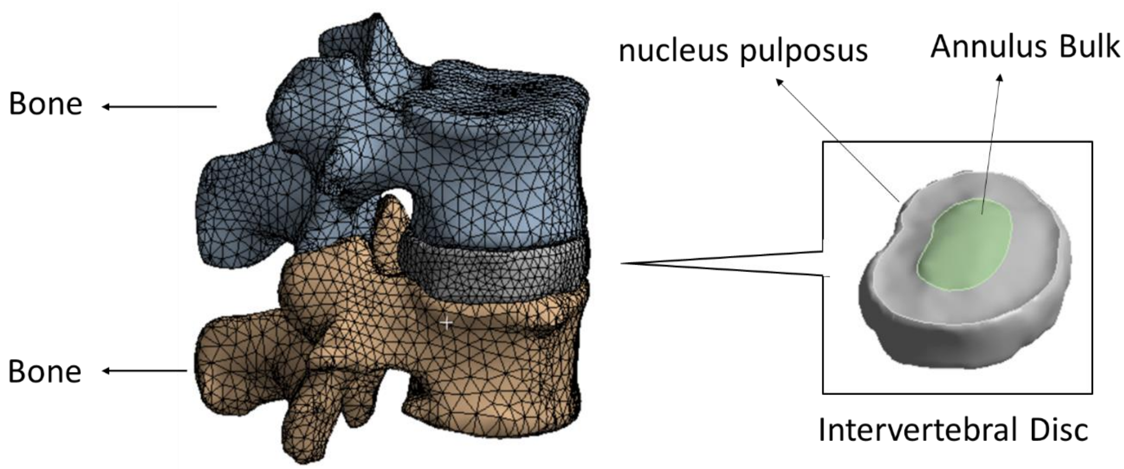

Figure 1. An illustration of different parts of the model

3. Define coupled pore-pressure-thermoelements. Insert commands to the nucleus and annulus part (Figure 2). These commands will change the element type to CPT217 and define new material properties for this part. The material properties for annulus bulk are elastic modulus $E = 2.5MPa$, Poisson’s ratios $v = 0.1$, and permeability $3 \times 10^{-4}\,mm^4/N/s$. For nucleus

pulposus, $E = 1.5MPa$ and other material properties are the same as the annulus. Bones have $E = 3500MPa$ and $v = 0.2$.

```
et, 102, 217          ! Create a new element type 217
keyo,102,12,1         ! Activate pore pressure DOF

esel,s,mat,,matid     ! Select elements in this part with matid
emodif,all,type,101 ! Change the element type to 101

E1=1.5                ! Young's modulus,MPa
new_mat_id = 12       ! Define a new material id
MP,EX,new_mat_id,E1           ! Set the material's Young's modulus
MP,NUXY,new_mat_id,0.1        ! Set the material's Poisson ratio

fpx1=3e-4             ! Fluid permeability

! Set the material's fluid permeability
TB,PM,new_mat_id,,,perm
TBDATA,1,fpx1,fpx1,fpx1

emodif,all,mat,new_mat_id    ! Change the material to new_mat_id

allsel, all
```

Figure 2. Commands to change the element type and define permeability

4. Define analysis settings and boundary conditions. Fix the x and y degree of freedom of the bonds and only allow it to move in the z-direction. Fix the bottom of the bones and apply a compression step load of $500N$ at the top (Figure 3). Define an analysis time of 426667s (approximately 5 days). Set the pressure degree of freedom of the outer nodes of the annulus as 0 by inserting a command to the "Analysis" (Figure 4).

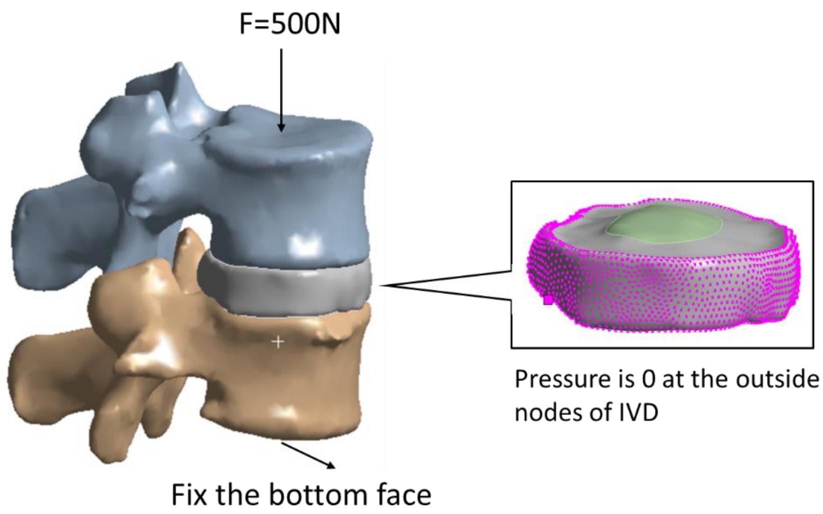

Figure 3. Boundary conditions

```
! boundary conditions
cmsel,s,pressure_zero_nodes, node        ! Select the nodes to apply zero
pressure
d,all,pres,0     ! Set the pressure to 0
allsel
```

Figure 4. Commands to set the pressure at IVD outside nodes to zero

5. Run the simulation. The result shows the creep behavior of IVD under constant loading. A contour plot of total deformation on the deformed shape is shown in Figure 5.

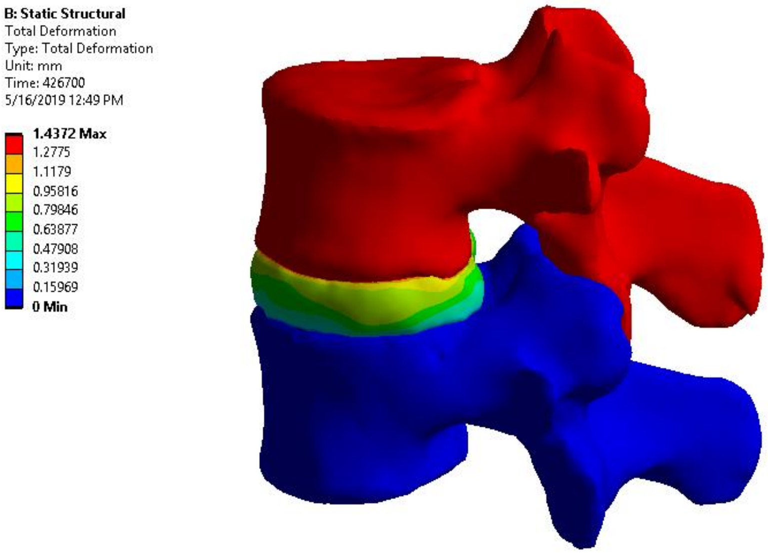

Figure 5. A contour plot of the deformation on the deformed shape

Summary

This document shows how to use Coupled pore-pressure-thermoelements (CPT217 elements and pore-pressure material) to model the biphasic behavior of IVD in Ansys. The material model can capture the creep behavior of IVD realistically.