Jan 10, 2020

Simulating colonic tissue mechanics using a structure-based material model in Abaqus

DOI
dx.doi.org/10.17504/protocols.io.wzeff3e

Bhavesh Patel[1]

[1]California Medical Innovations Institute

Bhavesh Patel

DOI: **https://dx.doi.org/10.17504/protocols.io.wzeff3e**

Protocol Citation: Bhavesh Patel 2020. Simulating colonic tissue mechanics using a structure-based material model in Abaqus. **protocols.io** **https://dx.doi.org/10.17504/protocols.io.wzeff3e**

Protocol status: Working
We use this protocol and it's working

Created: January 11, 2019

Last Modified: January 10, 2020

Protocol Integer ID: 19206

Keywords: Finite element, strain energy function, mechanical stress, simulating colonic tissue mechanics, colonic tissue mechanics, finite element software abaqus, available in the finite element software abaqus, material model in abaqus, such material model in abaqus, colon tissue, biomechanical simuation, based material model, material constitutive model, such material model, colon, tissue, abaqus

Abstract

The structure-based material constitutive model typically used for the colon tissue (DOI:10.1016/j.jmbbm.2017.08.031, DOI:10.1016/j.jmbbm.2013.02.016) is not readily available in the Finite Element software Abaqus, which is commonly used for biomechanical simuations. In this protocol, we provide step-by-step guidelines to use such material model in Abaqus using the User Subroutine implemented by the autor.

Guidelines

This protocol has been tested with Abaqus 6.13.

Troubleshooting

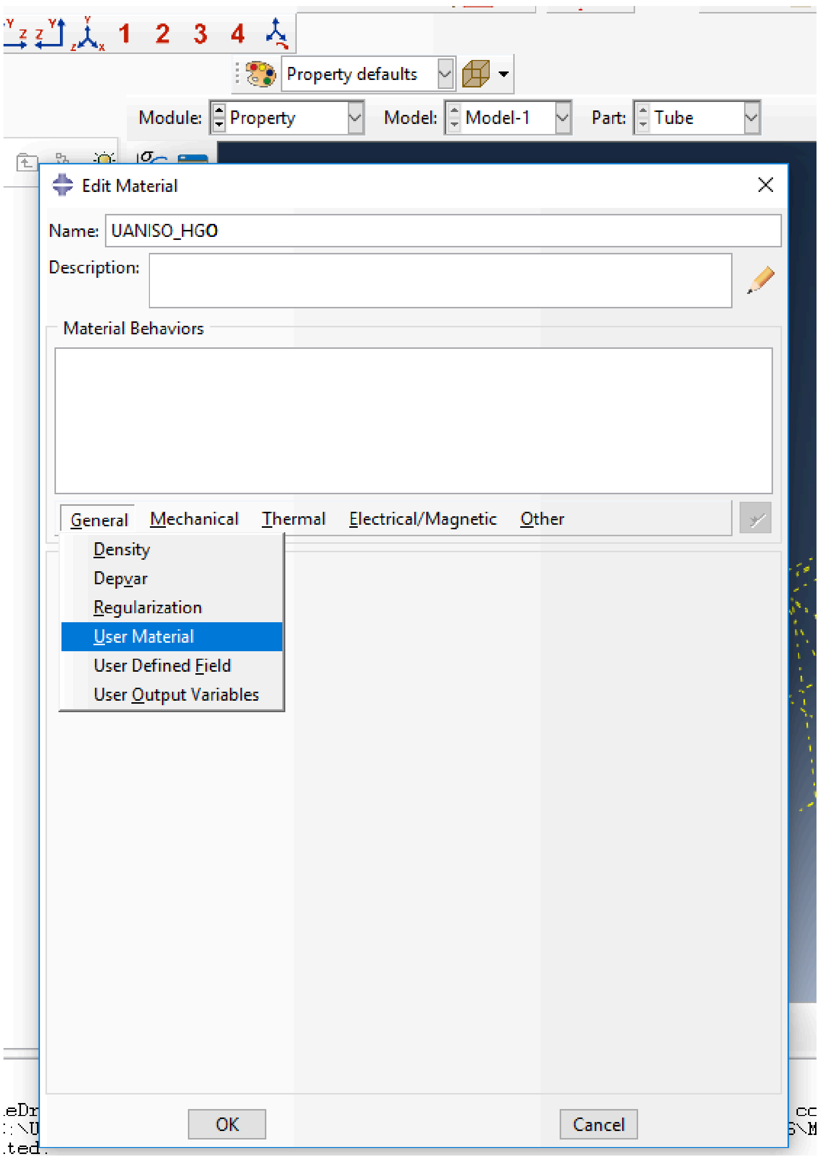
Before start

This protocols assume that you have Abaqus installed along with appropriate software to run Abaqus User Subroutines (e.g., **see this document**)

Assign user material model

1 Go the the "Property" module of Abaqus and click on "Create Material".

2 Enter "UANISO_HGO" for the name of the material in the "Edit Material" dialog box that opens, and select General →User Material.

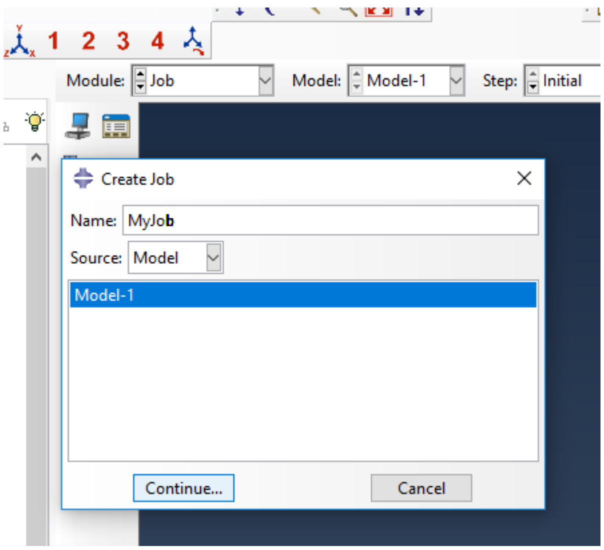


3 Click "OK" and assign the material to the desired sections of your model, as you would typically do with any material property.

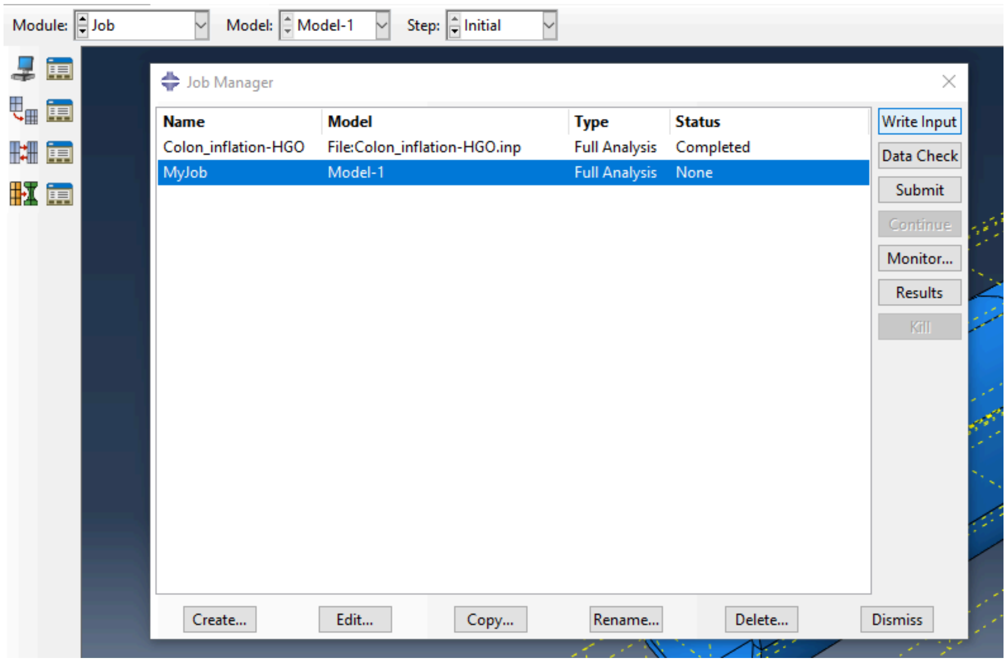
Generate input file

4 Once your model setup is completed (geometry definition, orientation assigment, meshing, etc.), go to the "Job" module and click on "Create Job".

5 Give a job name (e.g., MyJob) and select "Continue", then "OK" in the next dialog box.



6 Your job should be now listed in the "Job Manager" dialog box. Select it and click "Write Input". An input file (MyJob.inp) must have been created in your Work Directory.

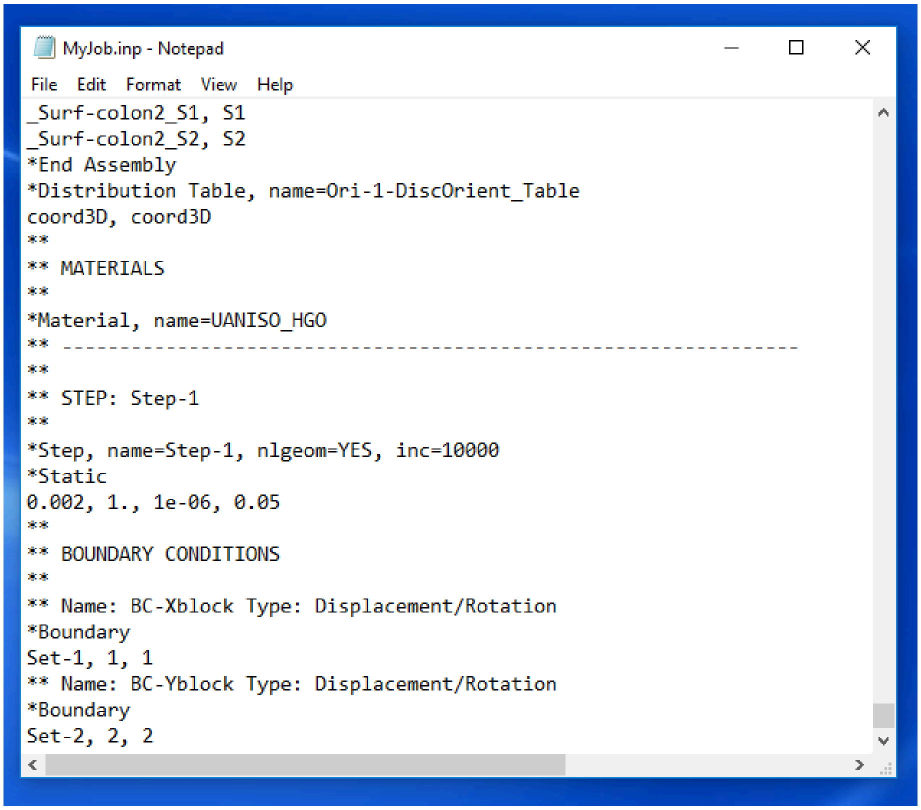


Specify material parameters in input file

7 The structure-based material constitutive model typically proposed for the colon is shown below. The values of your material parameters C_10, k1_l, k2_l, k1_s, and k2_s (pre-determined from mechanical testing or gathered from pre-published work) must be specified in the input file. To do so, open the input file with any text editor (e.g. Notepad).

$$\overline{W} = C_{10}(I_1 - 3) + \frac{k_1^l}{k_2^l}\left[e^{k_2^l(\lambda_z^2-1)^2} - 1\right] + \frac{k_1^s}{k_2^s}\left[e^{k_2^s(I_4^s-1)^2} - 1\right]$$

8 Go to the MATERIALS section of the file. You shoud see the material UANISO_HGO (defined in step 2) listed here.



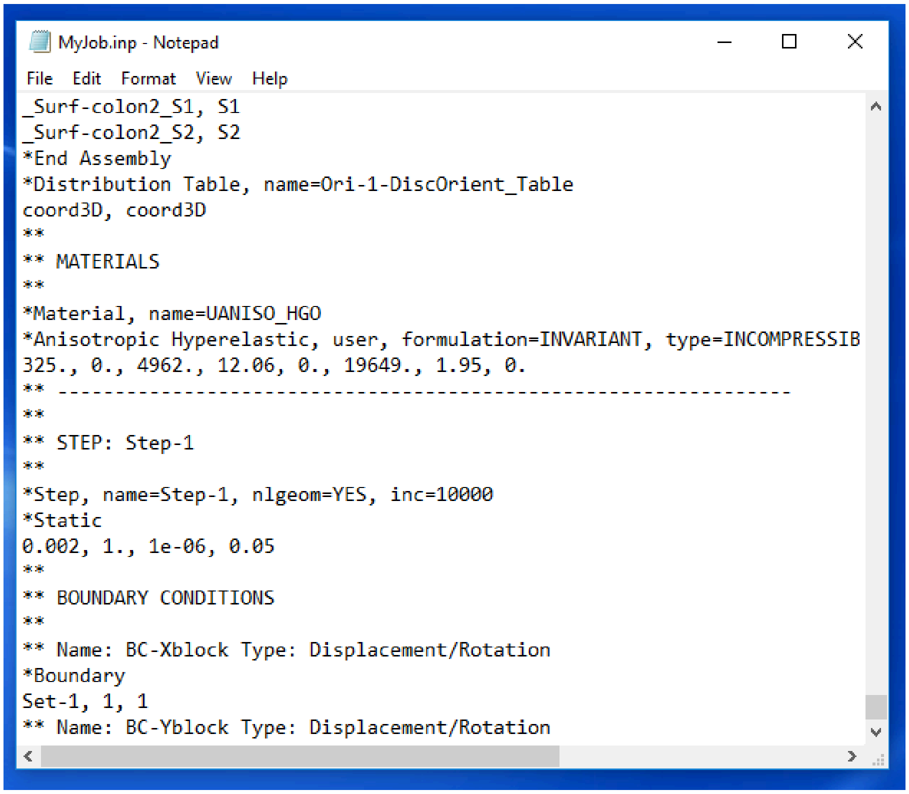
MyJob.inp - Notepad

File Edit Format View Help

```
_Surf-colon2_S1, S1
_Surf-colon2_S2, S2
*End Assembly
*Distribution Table, name=Ori-1-DiscOrient_Table
coord3D, coord3D
**
** MATERIALS
**
*Material, name=UANISO_HGO
** ----------------------------------------------------------------
**
** STEP: Step-1
**
*Step, name=Step-1, nlgeom=YES, inc=10000
*Static
0.002, 1., 1e-06, 0.05
**
** BOUNDARY CONDITIONS
**
** Name: BC-Xblock Type: Displacement/Rotation
*Boundary
Set-1, 1, 1
** Name: BC-Yblock Type: Displacement/Rotation
*Boundary
Set-2, 2, 2
```

9 Specify the value of material parameters material parameters by entering the following lines below the name of the material (copy/past this and replace material parameter names by the desired values):

∗Anisotropic Hyperelastic, user, formulation=INVARIANT, type=INCOMPRESSIBLE, local direction=3, properties=8
C_10, 0., k1_s., k2_s, 0., k1_l, k2_l, 0.




```
_Surf-colon2_S1, S1
_Surf-colon2_S2, S2
*End Assembly
*Distribution Table, name=Ori-1-DiscOrient_Table
coord3D, coord3D
**
** MATERIALS
**
*Material, name=UANISO_HGO
*Anisotropic Hyperelastic, user, formulation=INVARIANT, type=INCOMPRESSIB
325., 0., 4962., 12.06, 0., 19649., 1.95, 0.
** ----------------------------------------------------------------
**
** STEP: Step-1
**
*Step, name=Step-1, nlgeom=YES, inc=10000
*Static
0.002, 1., 1e-06, 0.05
**
** BOUNDARY CONDITIONS
**
** Name: BC-Xblock Type: Displacement/Rotation
*Boundary
Set-1, 1, 1
** Name: BC-Yblock Type: Displacement/Rotation
```

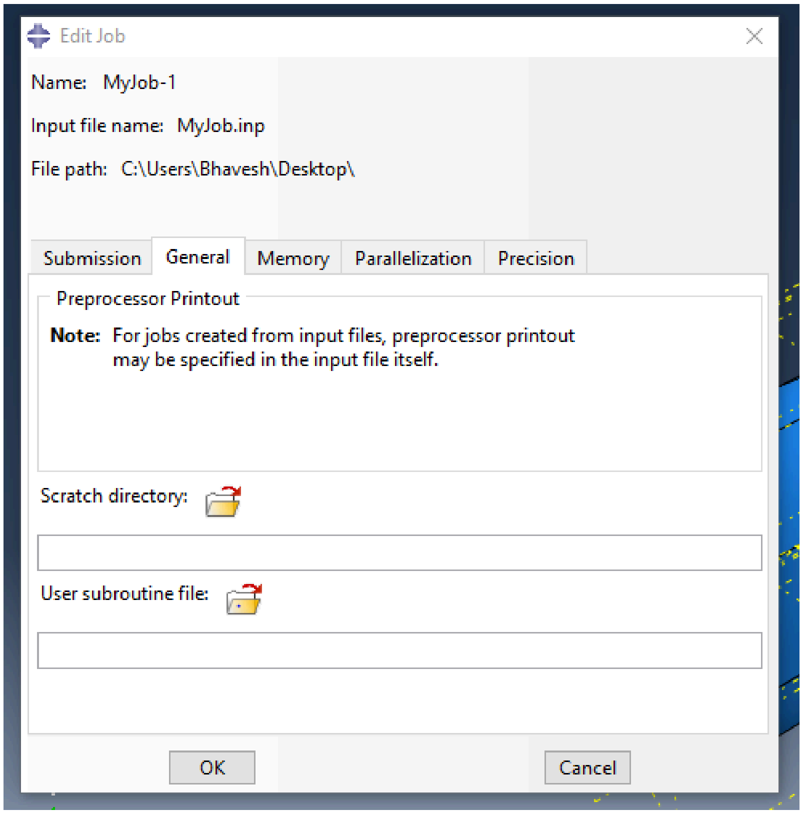
10 Save the input file and close it.

Run simulation

11 Go back to the "Job" module of Abaqus and create a new job.

12 Select "Input file" under "Source" and browse to the location of your input file then click "Continue".

13 Go to the "General" tab of the new dialog box "Edit Job". Under "User subroutine file", browse to the file "uanisohyper_inv_colon.for". Complete any other specification you may want to include and click "OK".



14 You can now select your job from the "Job Manager" dialog box and click on "Submit" to run your simulation.