

Jan 10, 2020

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

DOI

[dx.doi.org/10.17504/protocols.io.wzeff3e](https://dx.doi.org/10.17504/protocols.io.wzeff3e)

Bhavesh Patel<sup>1</sup>

<sup>1</sup>California Medical Innovations Institute



Bhavesh Patel

OPEN  ACCESS



DOI: [dx.doi.org/10.17504/protocols.io.wzeff3e](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) <https://dx.doi.org/10.17504/protocols.io.wzeff3e>

**License:** This is an open access protocol distributed under the terms of the [Creative Commons Attribution License](https://creativecommons.org/licenses/by/4.0/), which permits unrestricted use, distribution, and reproduction in any medium, provided the original author and source are credited

**Protocol status:** Working

**We use this protocol and it's working**

**Created:** January 10, 2019

**Last Modified:** January 10, 2020

**Protocol Integer ID:** 19206

**Keywords:** Finite element, strain energy function, mechanical stress

## 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 simulations. 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

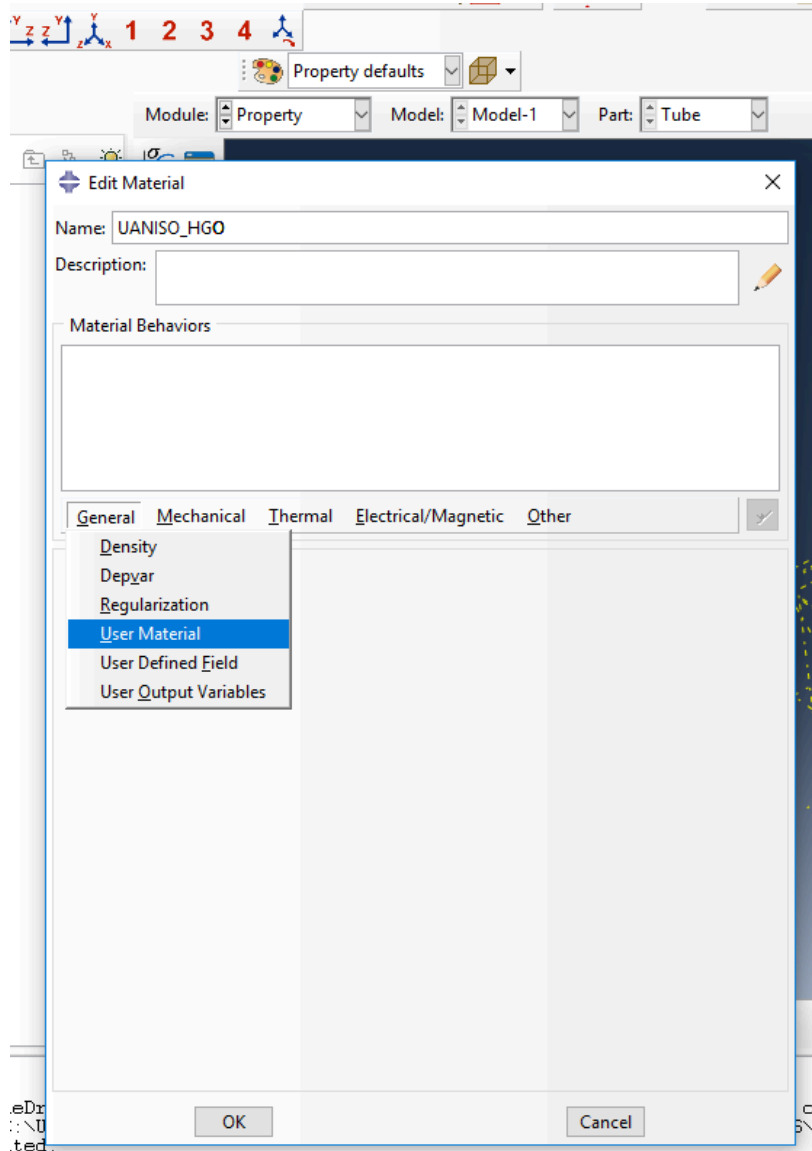
This protocol has been tested with Abaqus 6.13.

## 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

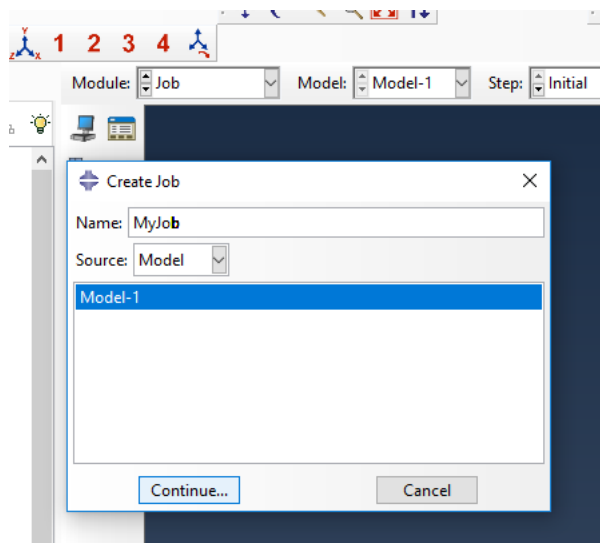
- 1 Go to the "Property" module of Abaqus and click on "Create Material".
- 2 Enter "UANISO\_HGO" for the name of the material in the "Edit Material" dialog box that opens, and select General → User Material.



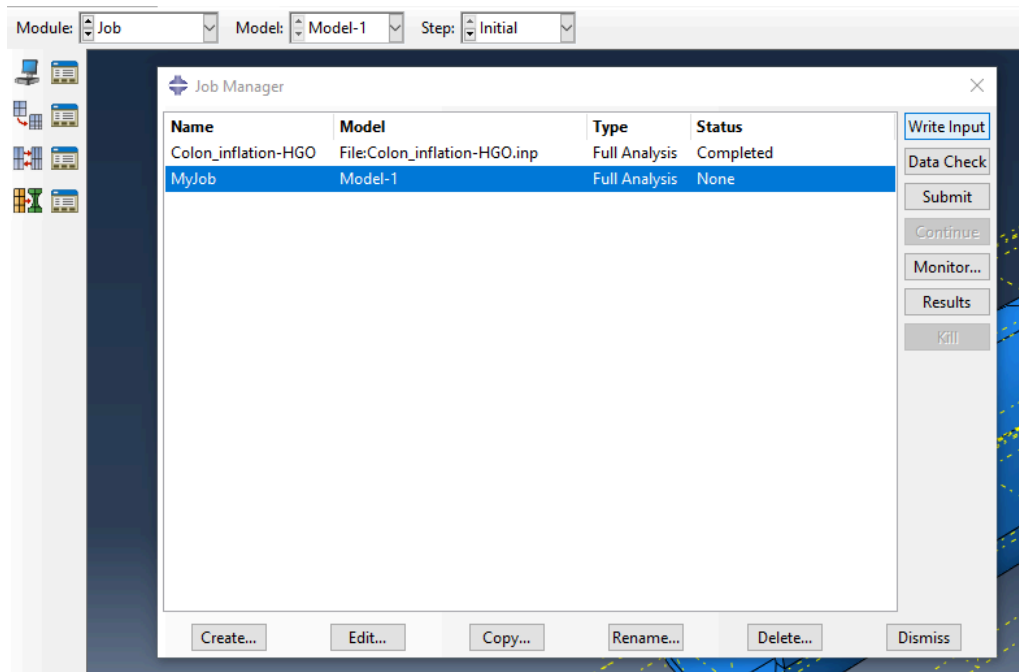
- Click "OK" and assign the material to the desired sections of your model, as you would typically do with any material property.

## Generate input file

- Once your model setup is completed (geometry definition, orientation assignment, meshing, etc.), go to the "Job" module and click on "Create Job".
- Give a job name (e.g., MyJob) and select "Continue", then "OK" in the next dialog box.



- 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.

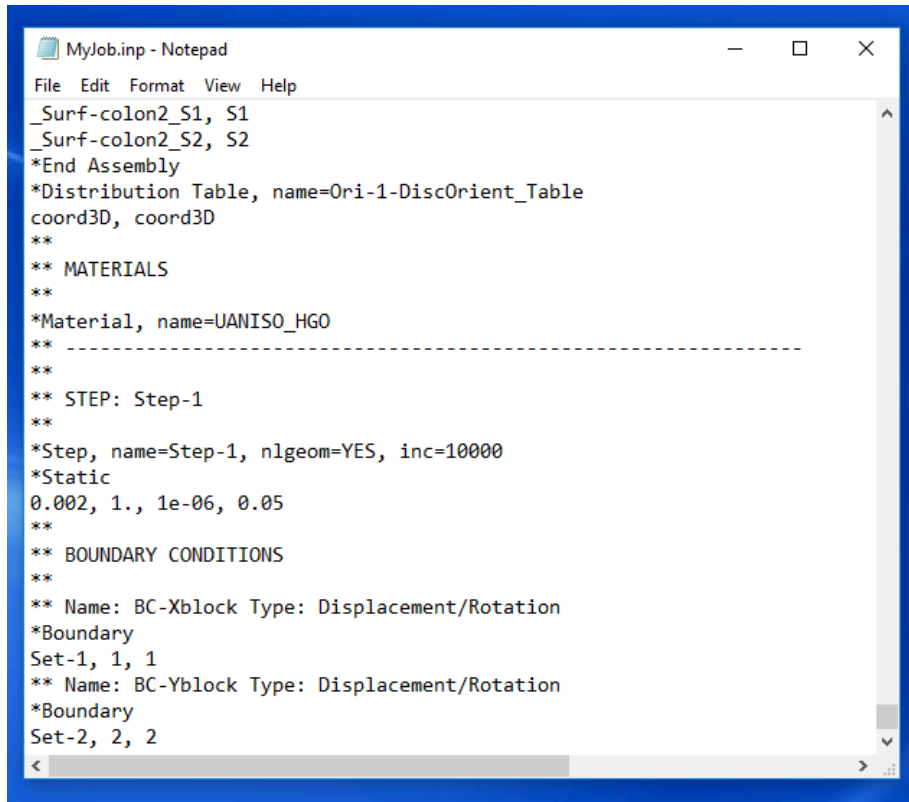


## Specify material parameters in input file

- 7 The structure-based material constitutive model typically proposed for the colon is shown below. The values of your material parameters  $C_{10}$ ,  $k_{1,l}$ ,  $k_{2,l}$ ,  $k_{1,s}$ , and  $k_{2,s}$  (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).

$$\bar{W} = C_{10}(I_1 - 3) + \frac{k_1^l}{k_2^l} \left[ e^{k_2^l (\lambda_2^2 - 1)^2} - 1 \right] + \frac{k_1^s}{k_2^s} \left[ e^{k_2^s (I_4 - 1)^2} - 1 \right]$$

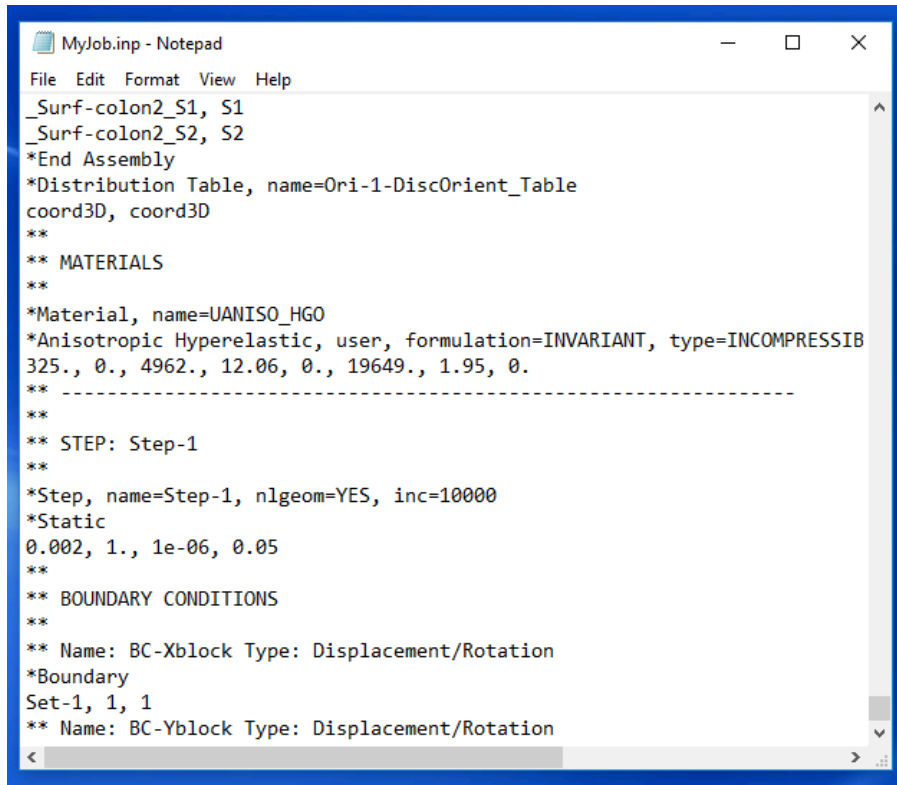
- 8 Go to the MATERIALS section of the file. You should see the material UANISO\_HGO (defined in step 2) listed here.



```
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.

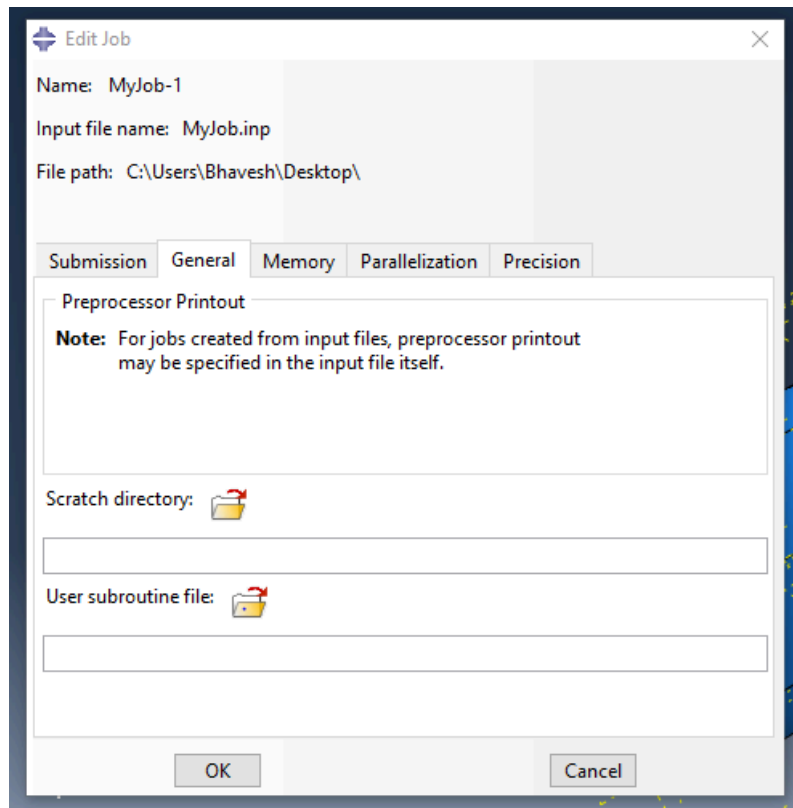


```
MyJob.inp - Notepad
File Edit Format View Help
_Surf-colon2_S1, S1
_Surf-colon2_S2, S2
*End Assembly
*Distribution Table, name=Ori-1-DiscOrient_Table
coord3D, coord3D
**
** MATERIALS
**
*Material, name=UANISO_HGO
*Anisotropic Hyperelastic, user, formulation=INVARIANT, type=INCOMPRESSIB
325., 0., 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.