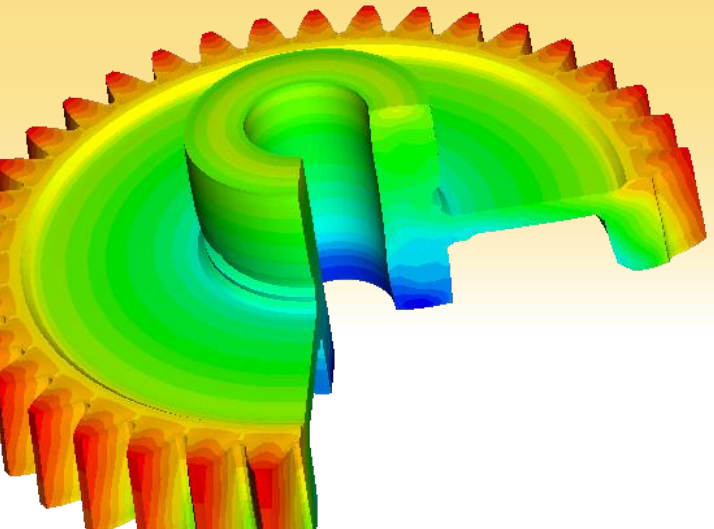
Dante 5.0 Tutorial

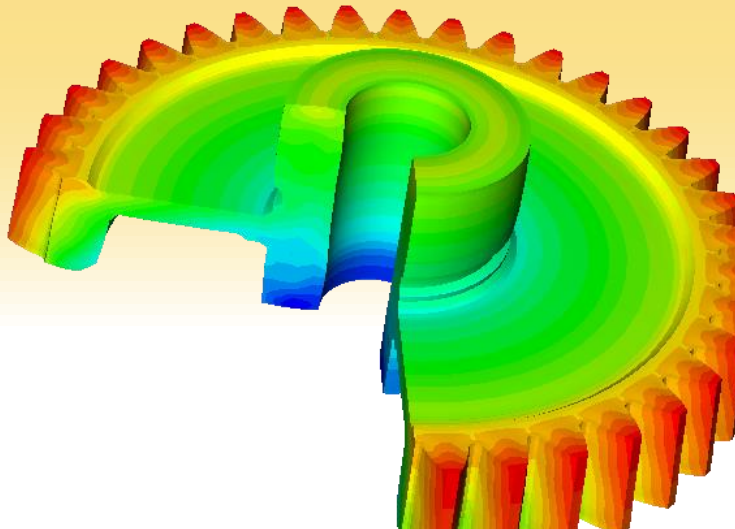
2D Axisymmetric Ring Model
Using Dante Model Builder Plug-In

Prepared By

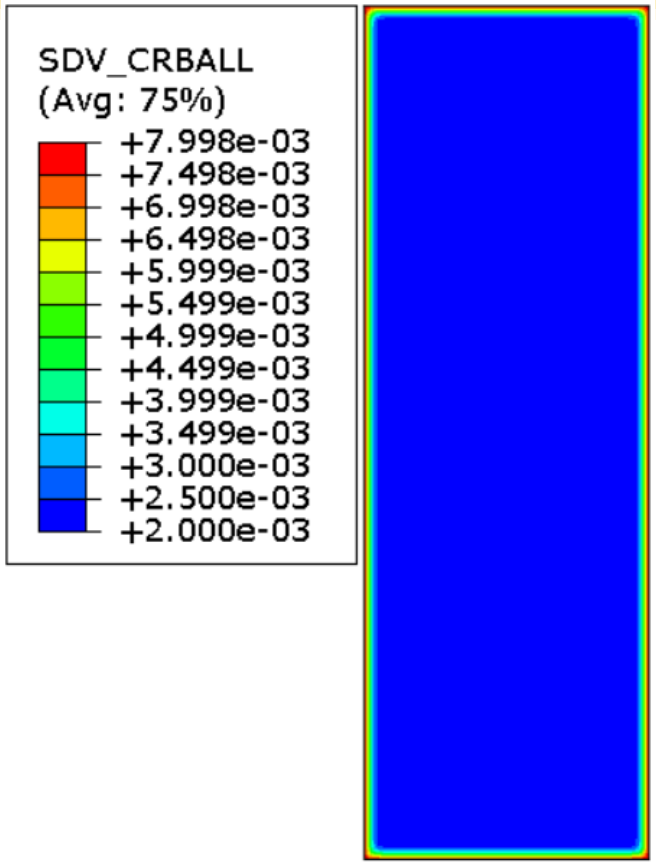
DANTE Solutions, Inc.

Cleveland Oh





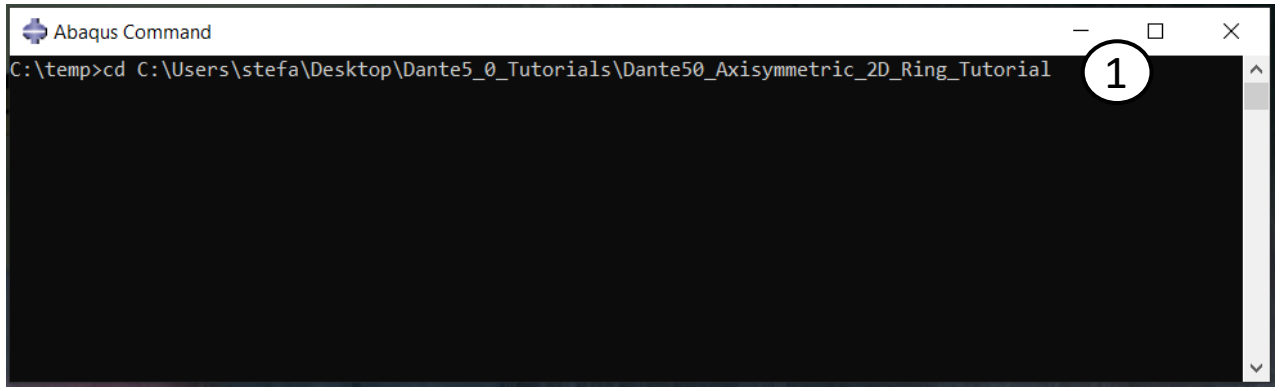
Carburization Model

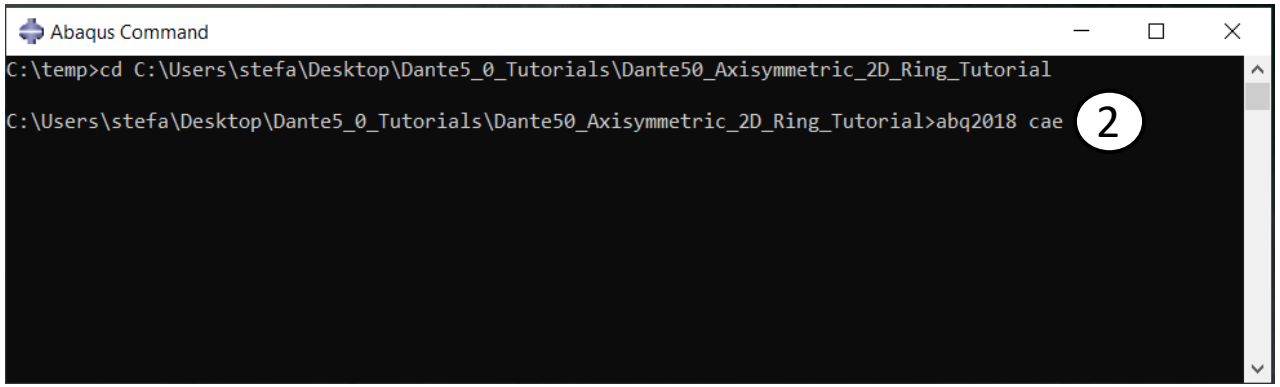


Preliminary Notes: The following tutorial will detail the complete set of steps required to set up model geometry, pre-process the model, along with running and post-processing of the model in the Abaqus CAE using the Dante Model Builder Plug-In. As with any model, be sure to save often. For further details on pre-processing and post-processing, please review the tutorial: ***Dante5_0_Tutorial_2D_Axisymmetric_Ring_Slice_CAE***, which builds this model using the CAE instead of the Plug-In interface.

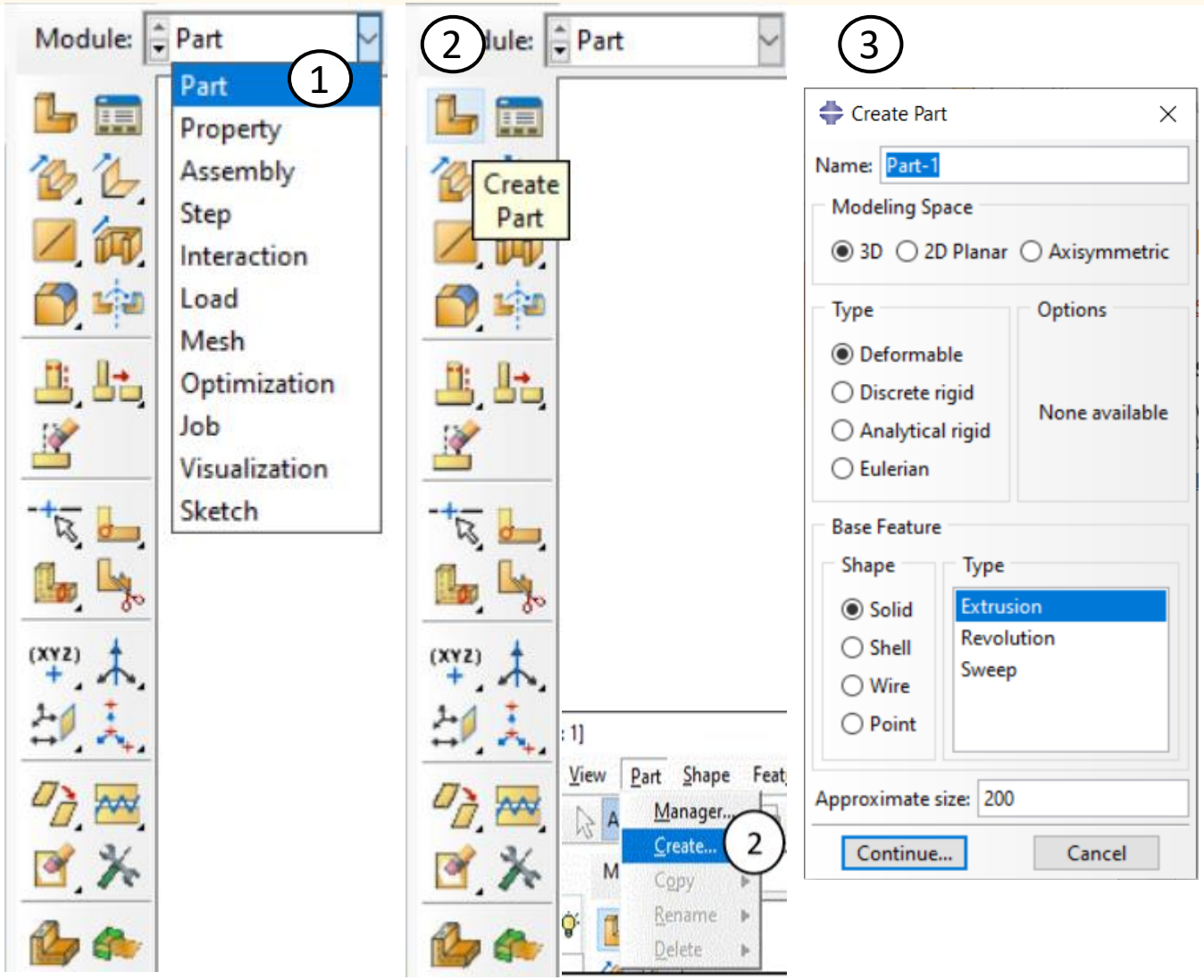
Step 1: Start Abaqus CAE

1. Open Abaqus Command and set the working directory by using the command ***"cd DirectoryPath"***
2. Start Abaqus CAE with the command ***"abqxxxx cae"*** into the command window
   - xxxx represents the version of Abaqus to be opened
3. Click ***With Standard/Explicit Model*** to start a new model

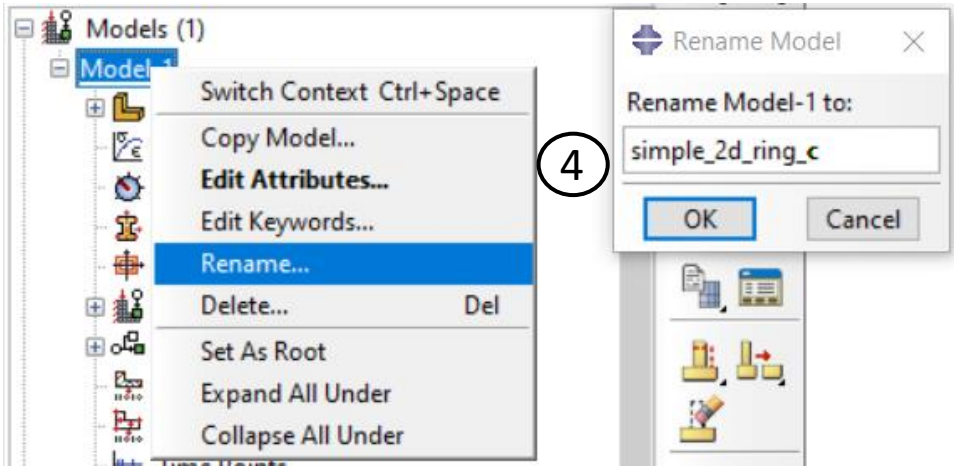




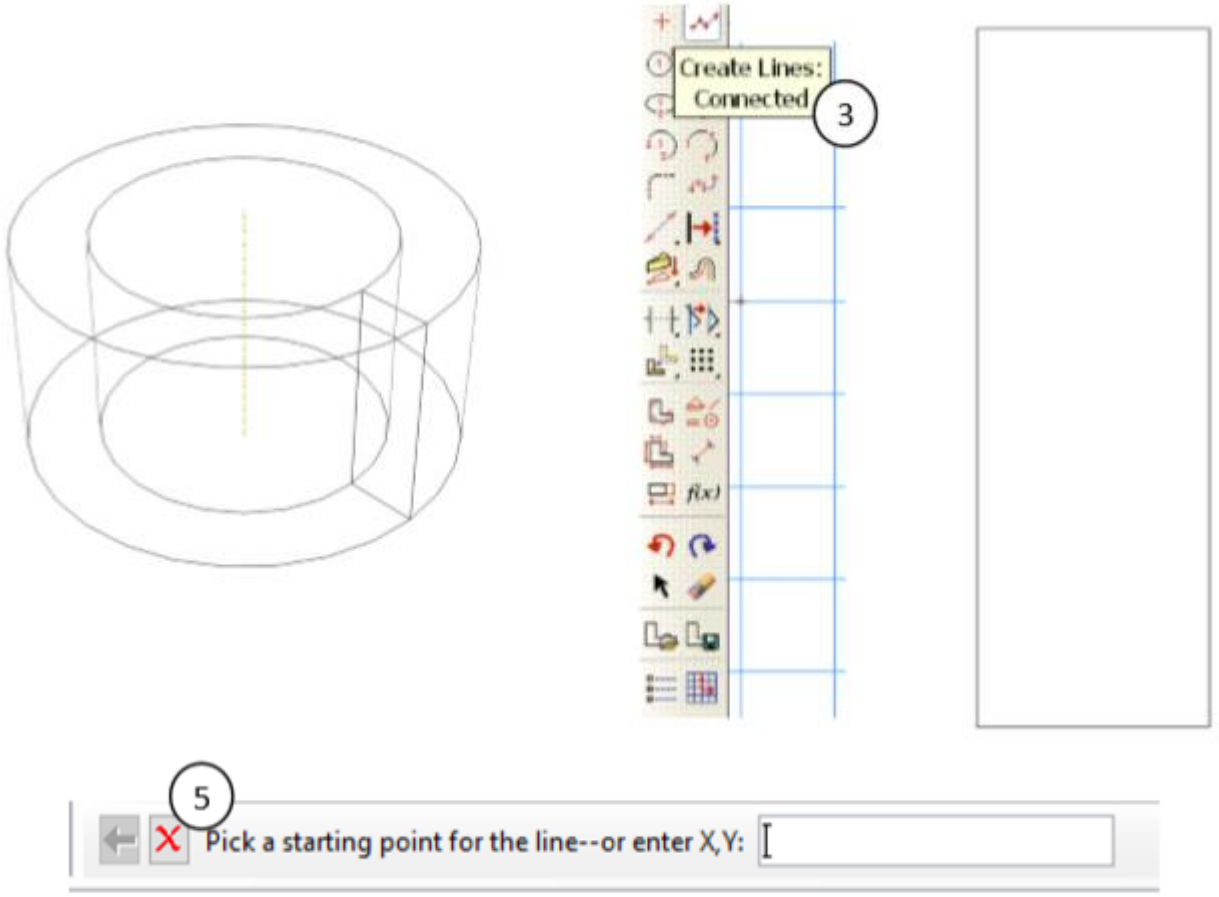
Step 2: Creating a Part



1. Under ***Module*** select Part to create a geometry
2. Click the ***Create Part*** icon ***or Part → Create*** in the dropdown menu at the top
3. Rename the part ***simple_2d_ring_c*** and ***select Axisymmetric, Deformable, Shell***, and set the approximate size to ***200*** then click continue
4. The model can also be renamed to reflect model type

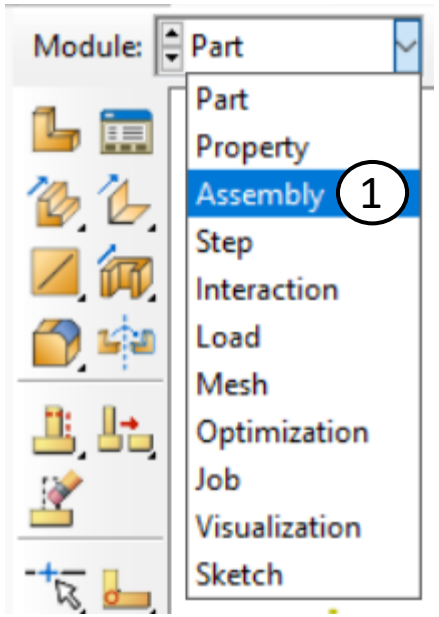


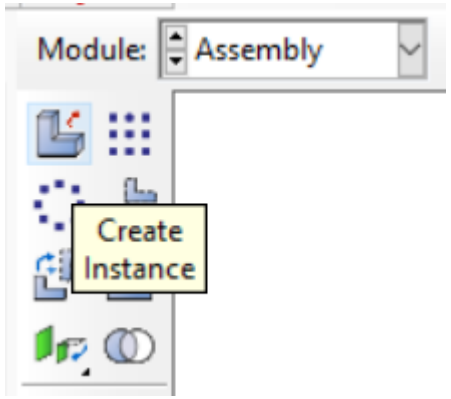
Step 2b: Creating a Part

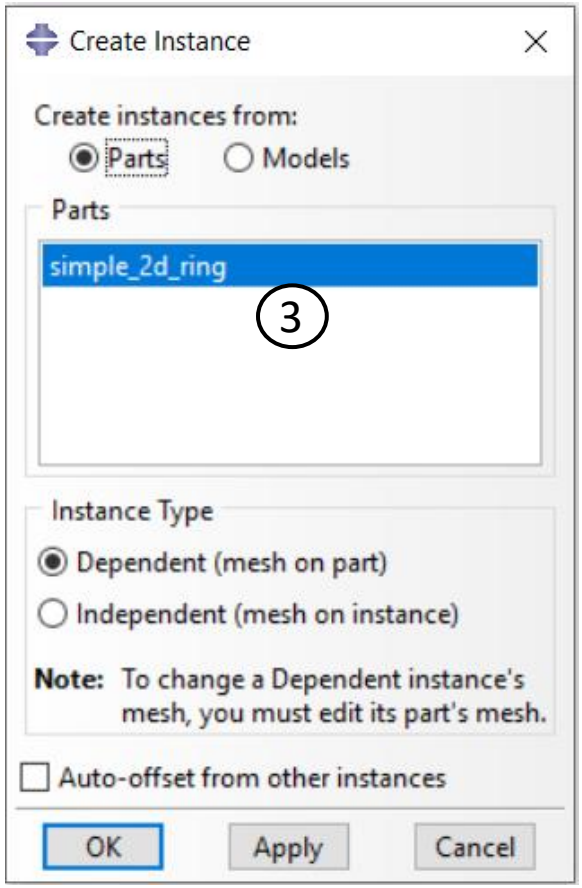


1. The ring has an inner diameter of 80 mm, outer diameter of 120 mm, and a height of 60 mm
2. An axisymmetric rectangular cross section will be used to model the ring
3. Select the ***Created Lines: Connected*** tool and type in (40,0), (60,0), (60,60), and (40,60)
4. Connect the last side by using your mouse and selecting the point (40,0)
5. Select the red ***X*** to exit the tool and press ***Done***

Step 3: Create an Instance

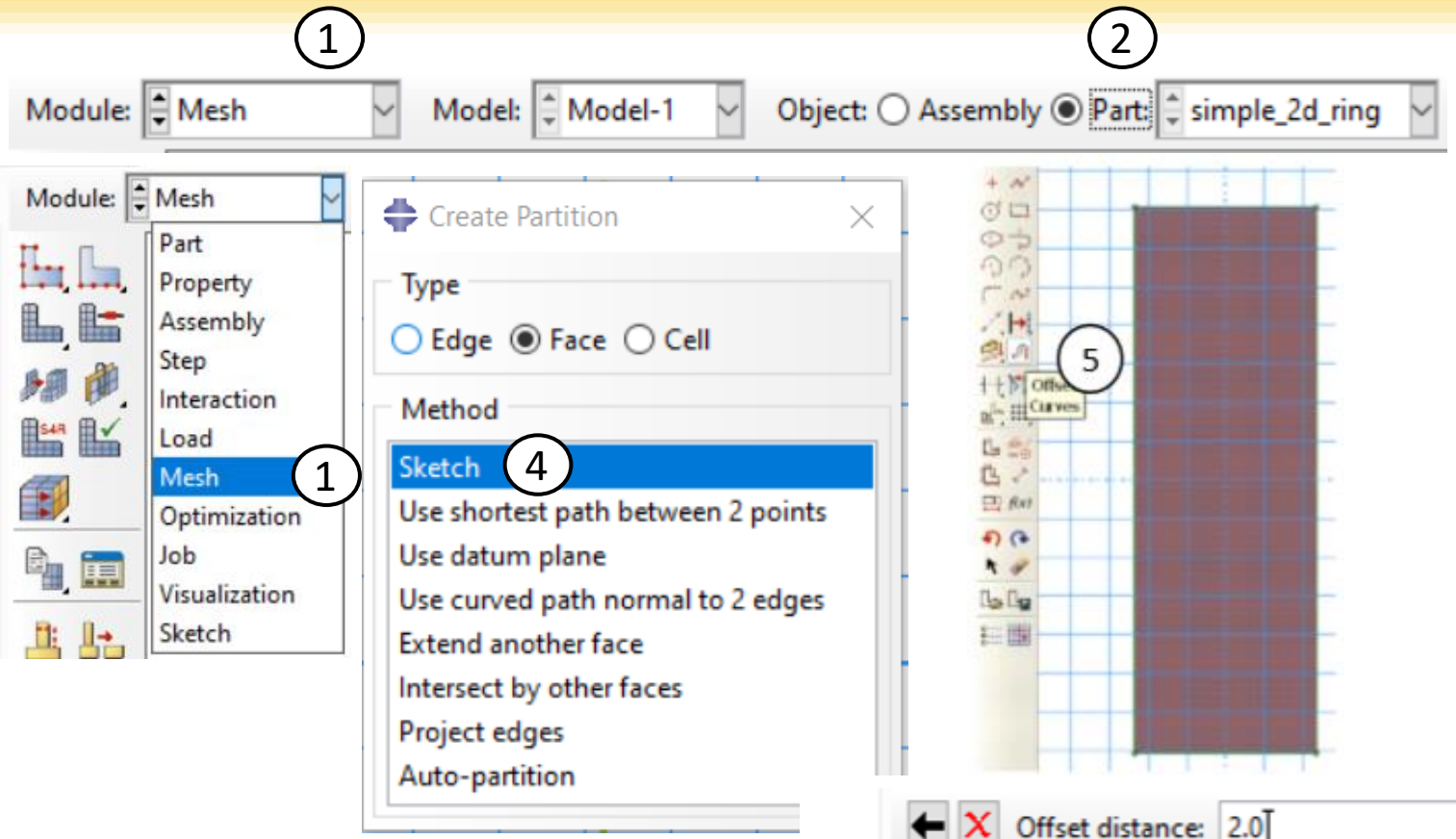


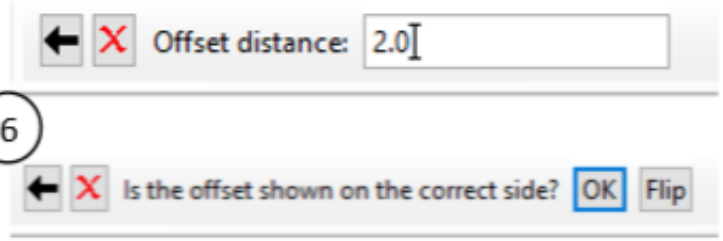


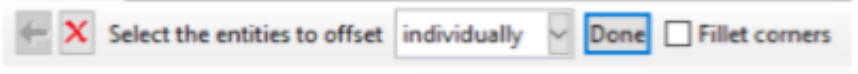


1. Under ***Module***, select ***Assembly*** to create the instance
2. Click the ***Create Instance*** icon or select ***Instance → Create***
3. Select the part to be instanced and check ***Dependent (mesh on part)***
4. Click ***OK*** and the part should turn blue

Step 4: Meshing the Part

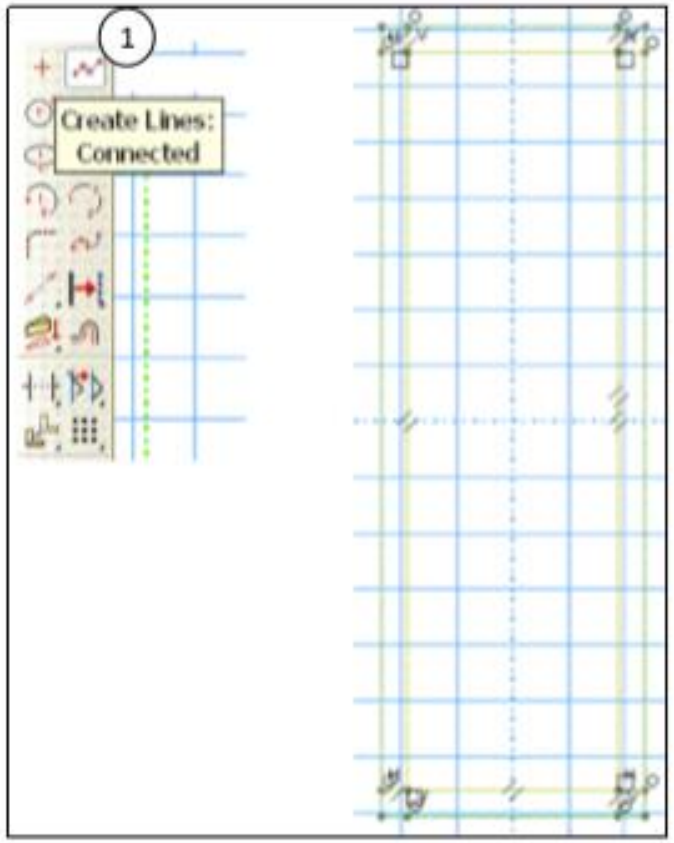


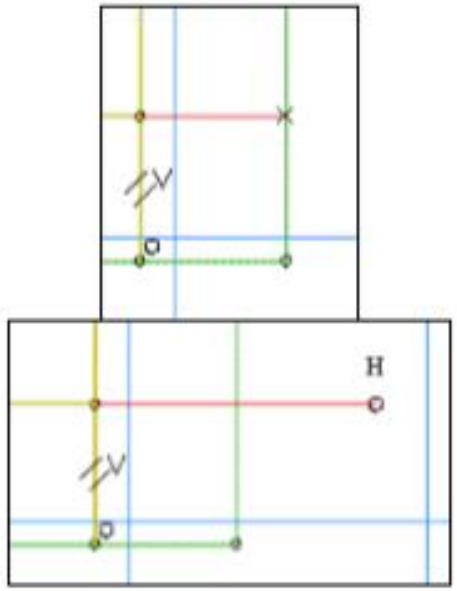


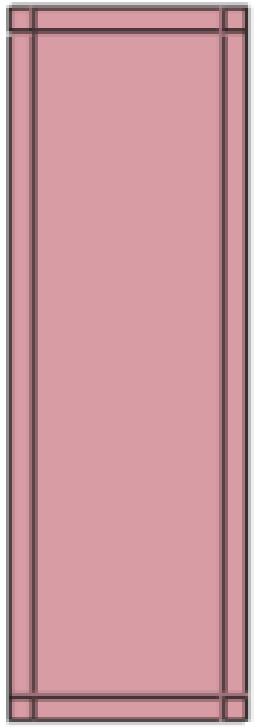


1. Under ***Module***, select ***Mesh*** to mesh part
2. Check ***Part*** instead of ***Assembly***
3. Select ***Tools → Partition*** from the dropdown menu
4. Check ***Face*** under Type and select ***Sketch*** under Method
5. Select ***Offset Curves***, select all sides, then click ***Done***
6. Set the Offset Distance to ***2***
   - Click ***Flip*** if partition lines are on the outside of the part
   - Click ***OK*** then ***Done*** when the offset lines are complete

Step 4b: Meshing the Part

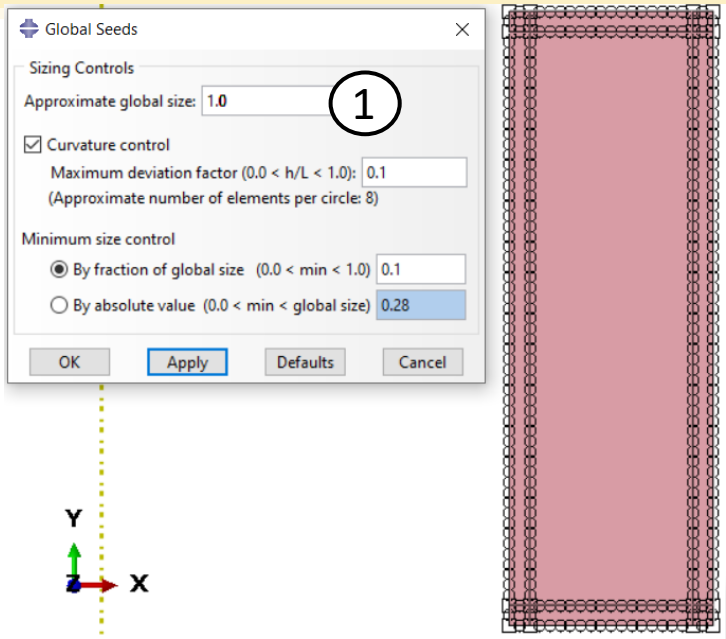




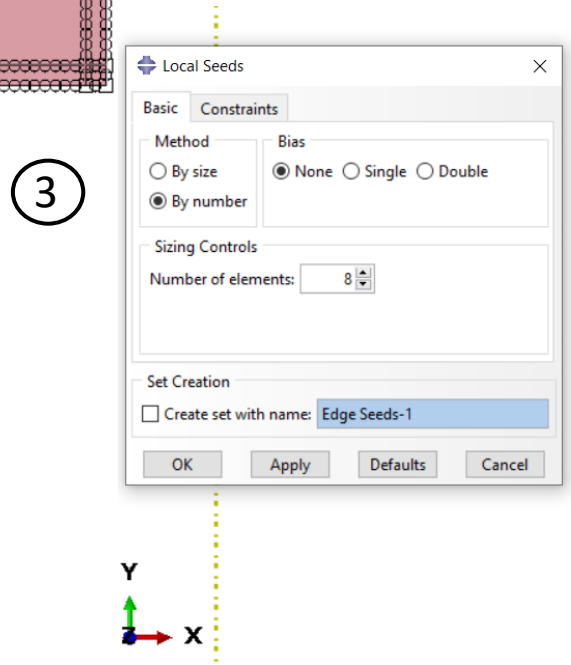


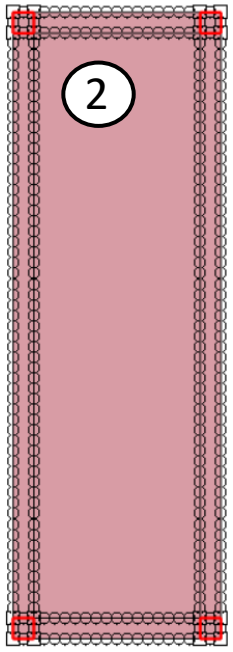
1. Select ***Create Lines: Connected*** to extend the partition lines in each corner to create squares and click the red X when complete
    - When creating a line, the “x”, “H”, “V” means the line is perfectly horizontal or vertical as seen in the images to the left
2. Click ***Done*** when the partition is complete
    - The final partition should look like the adjacent image

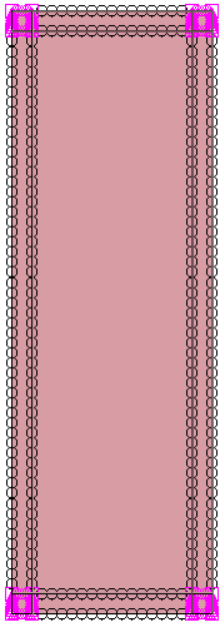
Step 4c: Meshing the Part



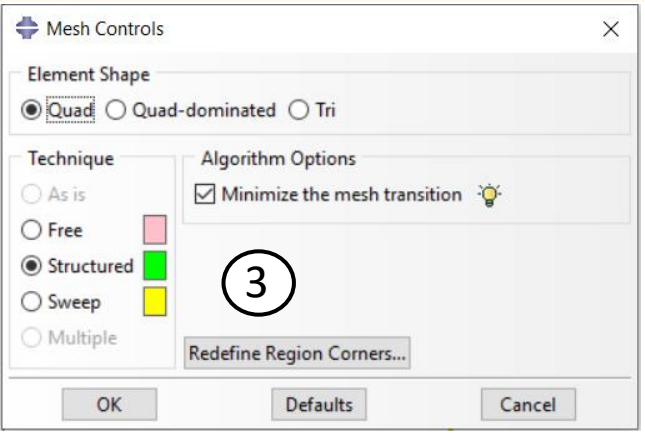
1. In the dropdown menu, select ***Seed → Part*** then set the ***Approximate global size*** to ***1.0*** and click ok
2. In the dropdown menu, now select ***Seed → Edges*** then select each edge of the four squares created during partitioning using ***Shift + Click*** and click ***OK*** when complete
3. In ***Method***, set the option to ***By Number*** and select ***8*** for ***Number of elements*** then click ***OK***
4. The final seeding can be seen in the image to the bottom right

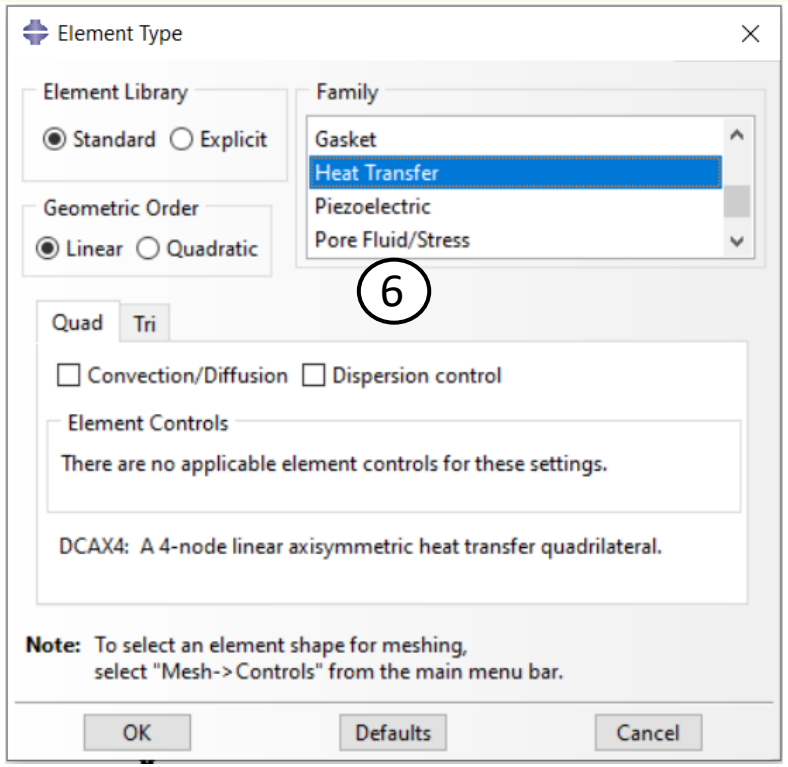


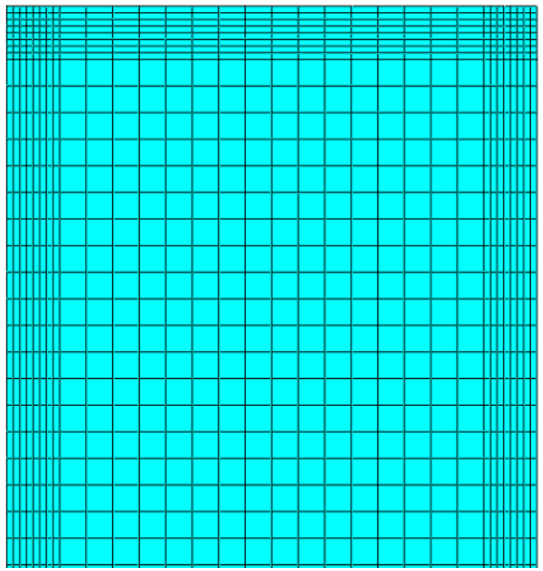




Step 4d: Meshing the Part







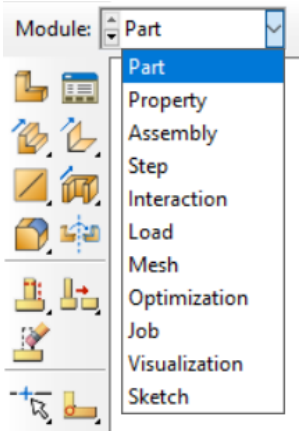
1. In the pulldown menu, select ***Mesh → Controls***
2. Box select the entire geometry then click ***Done***
3. Select ***Quad*** under Element Shape then select ***Structured*** under Technique and click ***OK*** when complete
4. In the dropdown menu, select ***Mesh → Element Type***
5. Box select the entire geometry then click ***Done***
6. Under ***Family***, select ***Heat Transfer***, keep the default settings and click ***OK*** when complete
7. In the dropdown menu, select ***Mesh → Part*** and click ***Yes*** to mesh the part
   - There should be a total of 2304 elements

Step 5: Model Continuation

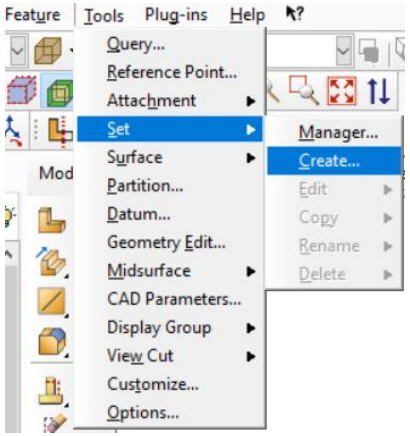
- At this point, the following steps have been completed:
  - Geometry has been defined
  - Instance has been created
  - Partitioning completed
  - Geometry has been meshed
- To complete this model, the Dante Model Builder Plug-In may be used
- Before doing so, surfaces and sets should be defined for initial/boundary conditions and interactions
- However they can be defined at any time when using the Plug-In interface
- Typically, it is best to layout process steps and boundary conditions prior to model setup to better understand how to configure the process in the simulation
- This carburization model will include a single step to model the diffusion of carbon into the ring while in a furnace:
  - Carburization, 10620 seconds total time
  - Material: 5120 carburized to near 0.8% Carbon content by weight at the surface

Step 6: Creating Preliminary Sets, Part Element Set

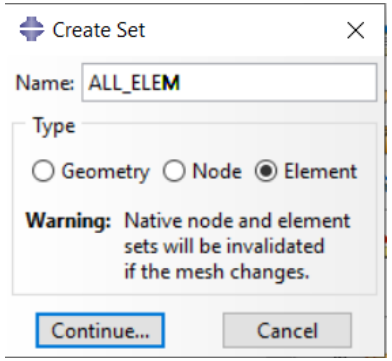
- With the mesh defined, sets and surfaces based on the mesh can be defined to ascribe boundary conditions to the model
- These boundary conditions will be created inside of the Dante Model Builder Plug-In
- Sets and surfaces must be created in the Abaqus CAE and they will appear in the Plug-In automatically to be selected
- First define sets of elements to ascribe material properties



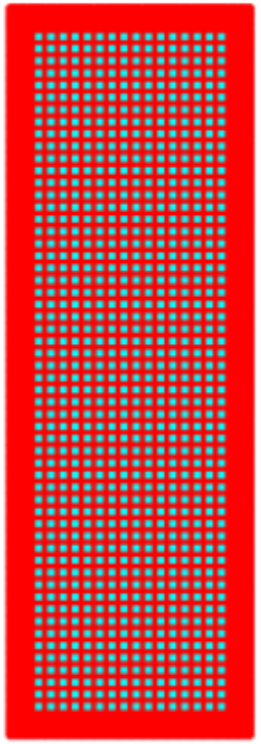
1



2



3

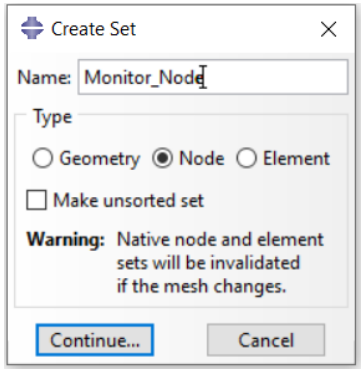


4

1. Go to the ***Part*** module
2. Under ***Tools → Set → Create***
3. Give a name, select ***Element***, and click ***Continue***
4. Box select all the elements and click ***Done***
   - The entire cylinder is made of one material so all elements in the part are selected

Step 6b: Creating Preliminary Sets, Monitor Node Set

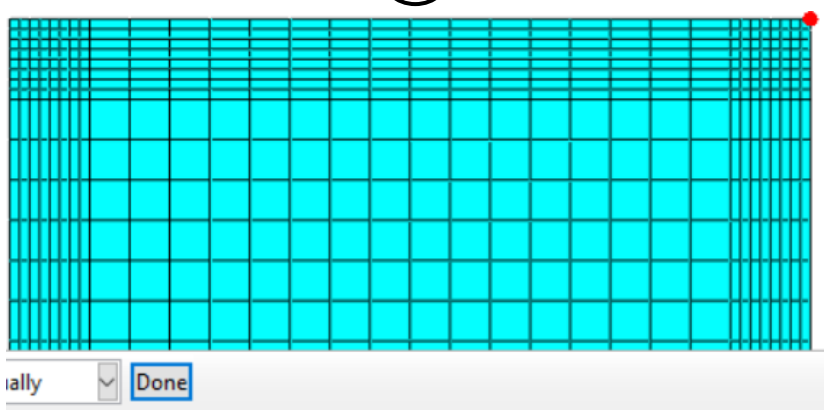
- A monitor node is a node selected to allow for monitoring of the simulation job during computation
- Any node can be chosen from the mesh, depending on the area to be monitored
- The monitor node will be created in the Dante Model Builder Plug-In



2

1. In the ***Assembly*** module, create a node set by going to ***Tools → Set → Create***
2. Rename the set aptly and select ***Node*** and ***Continue***
3. Select a node on the part and click ***Done***

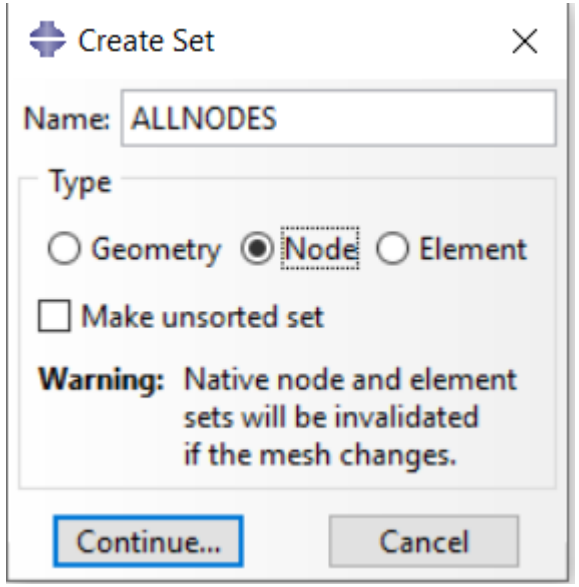
3

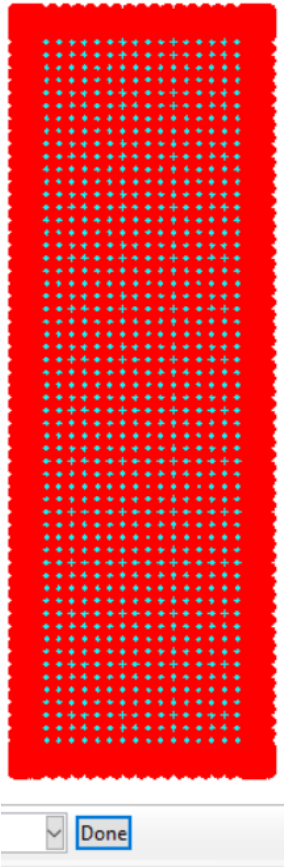


Step 6c: Preliminary Sets, All Nodes

- Field variable values, such as carbon content and temperature, are applied to the nodes of the model
- So a node set must be defined containing all the nodes in the model to which these field values are to be applied

1. In the ***Assembly*** module, create a node set by going through ***Tools → Set → Create***
2. Rename the set 'ALLNODES' and select ***Node*** and ***Continue***
3. Box select all the nodes in the model and click ***Done***

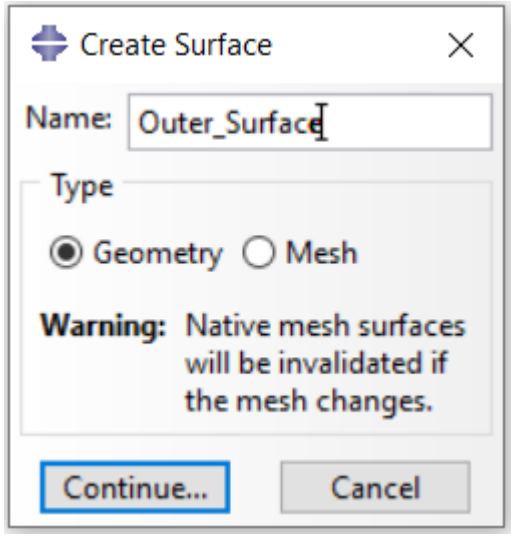


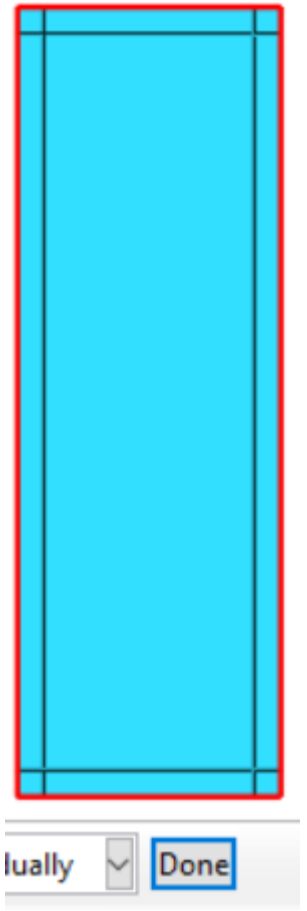


Step 6d: Preliminary Surfaces

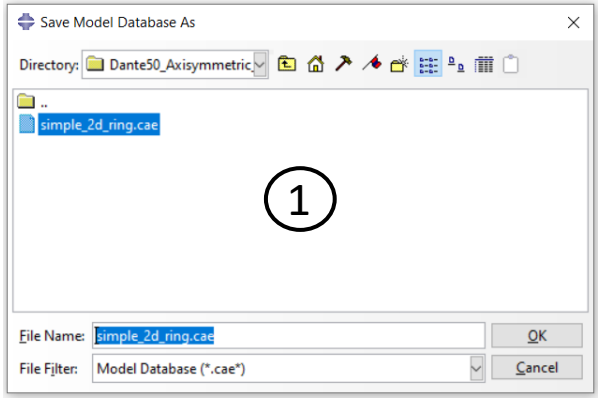
- Heat transfer coefficient boundary conditions are applied on surfaces, so it is best to visualize what heat transfer conditions are like on the surfaces of the components in question
- In this ring slice, the entire outer surface is exposed to the furnace atmosphere so the surface will be the boundaries of the geometry

1. In the ***Assembly*** module, create a surface by going to ***Tools → Surface → Create***
2. Rename the surfaces 'Outer_Surface' and select ***Geometry*** and ***Continue***
3. Box select all the surfaces in the model and click ***Done***

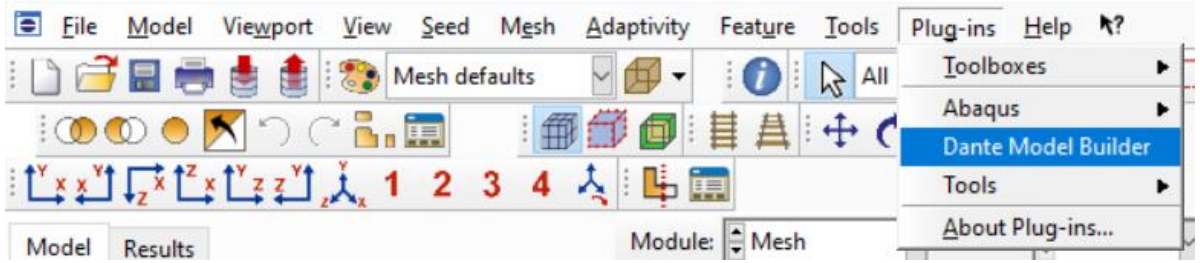


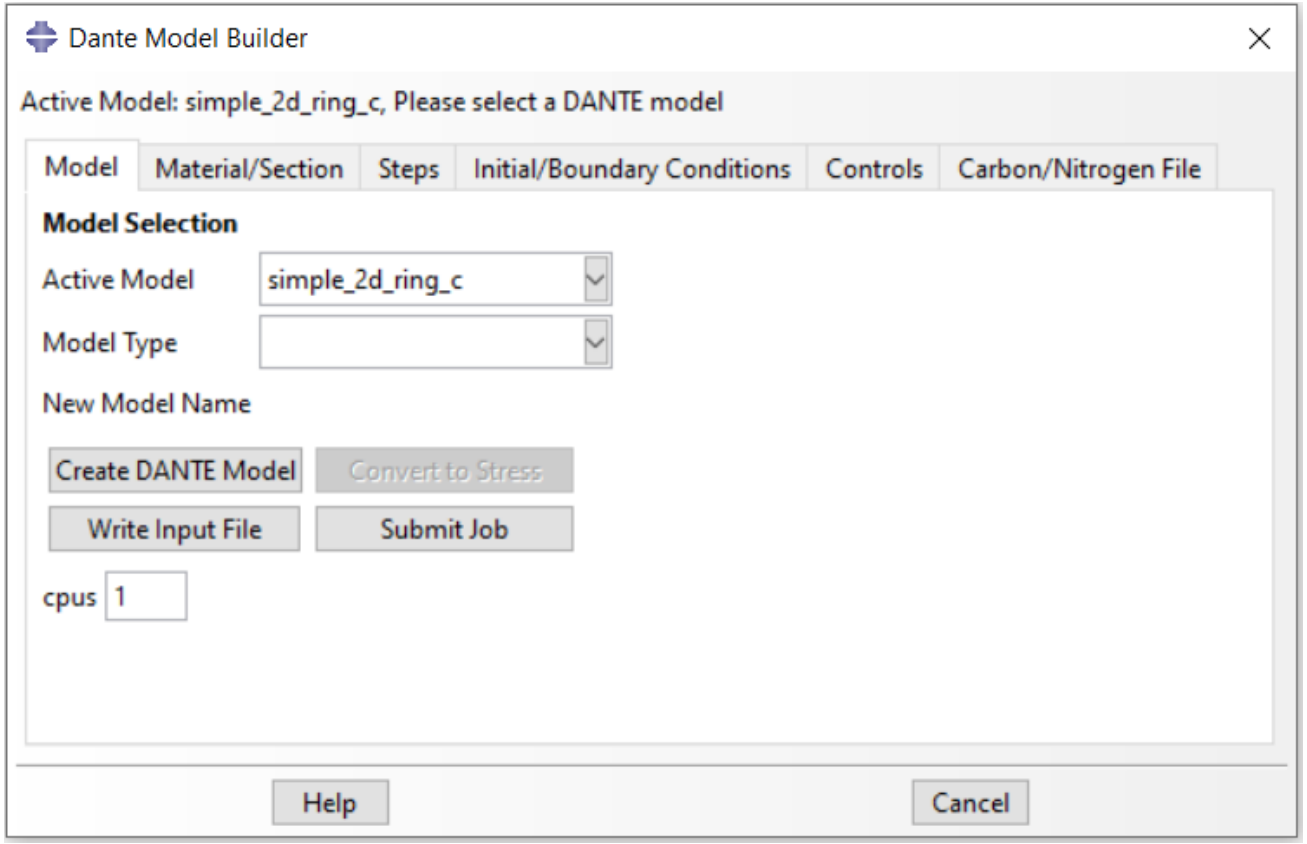


Step 7: Launching the Dante Model Builder Plug-In



1. Be sure to have saved the cae model under ***File***
2. Launch the Dante Model Builder Plug-In from ***Plug-ins →Dante Model Builder***
3. The following graphical user interface (gui) will appear

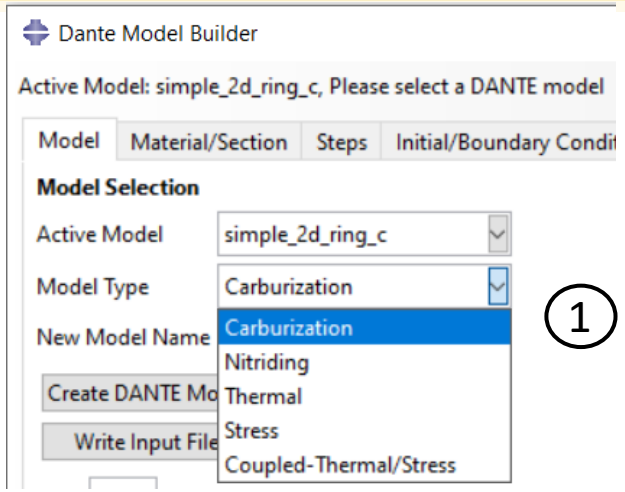


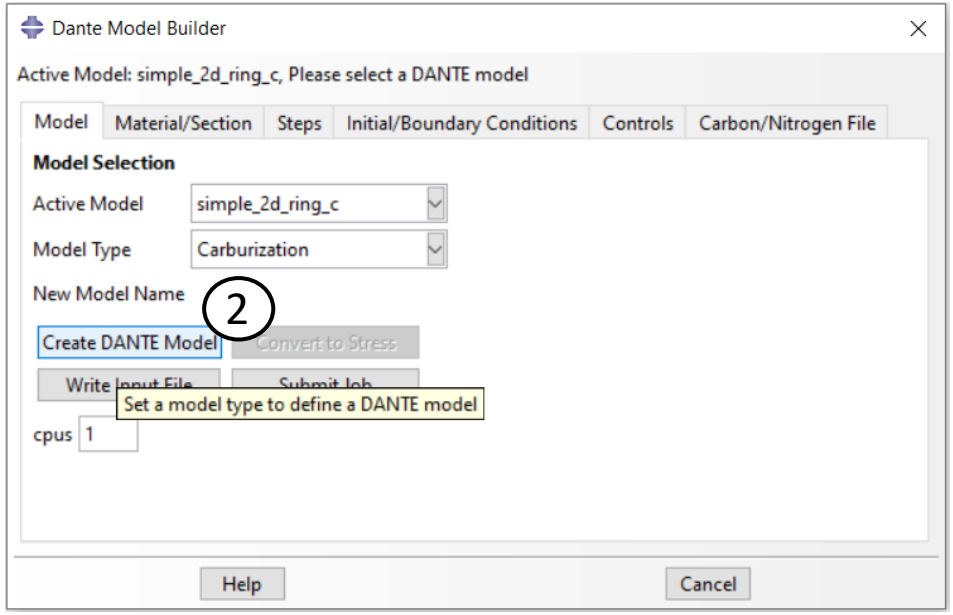


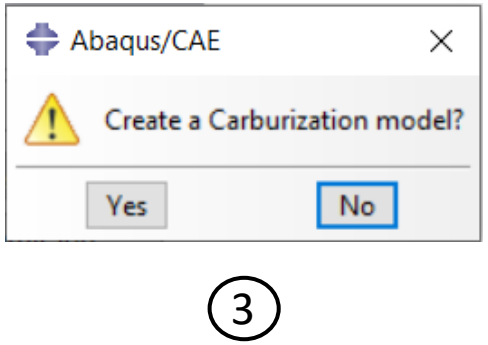
- The Dante Model Builder packages together all aspects of the model set up stages and inserts all the proper information relevant to interface the DANTE user subroutines into the model

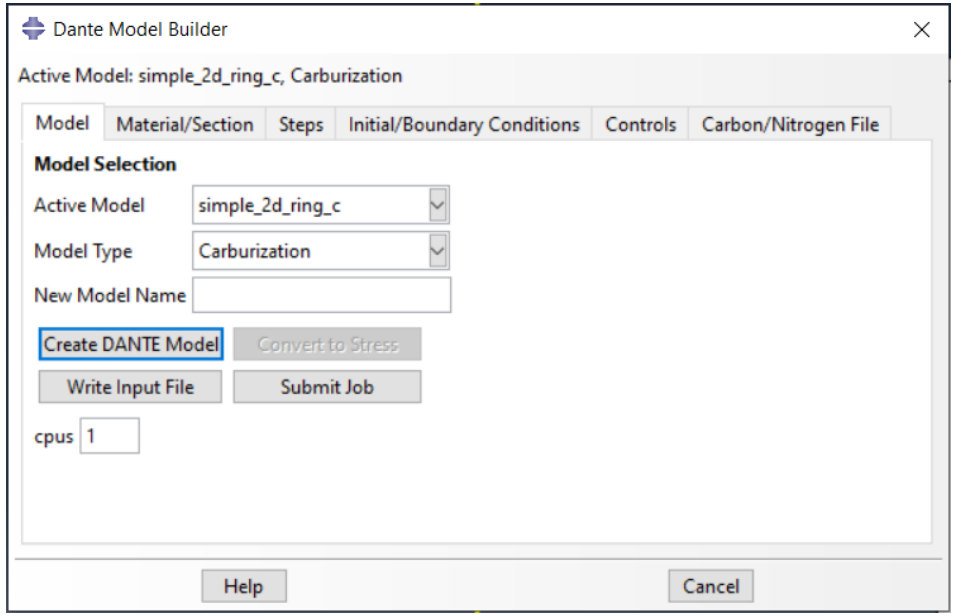
Step 8: Setting Up the Dante Model

1. To set up a carburization model, select the dropdown menu next to ***Model Type*** and select ***Carburization***
2. Click ***Create DANTE Model*** to apply the change
   - This will prompt a warning dialog asking to confirm model type
3. Click ***Yes*** on this dialog to create model type
   - This will set the Dante model type for the model and the Active Model label at the top of the gui should update to reflect the change

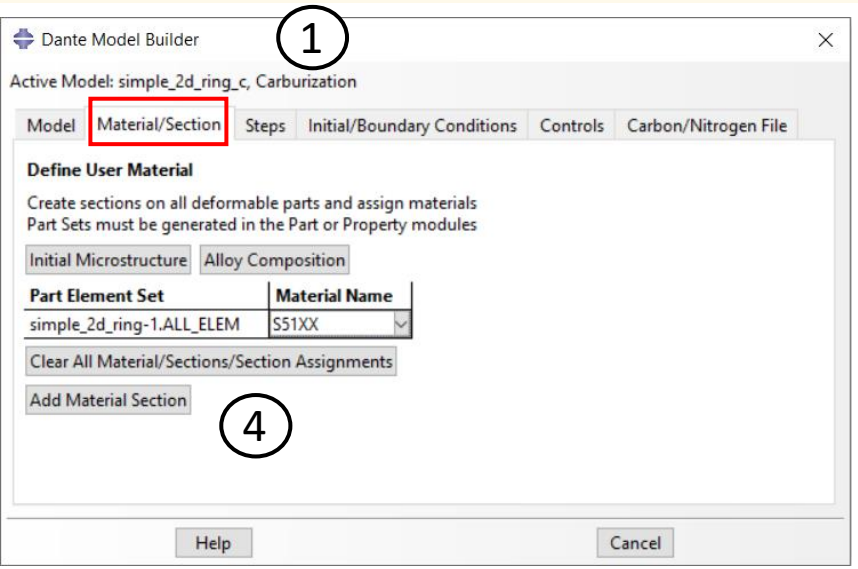


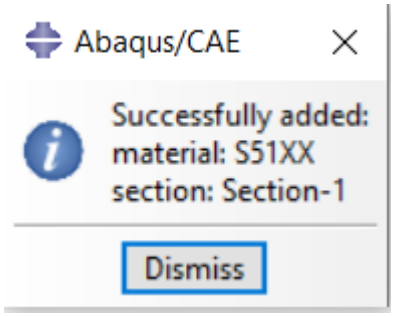






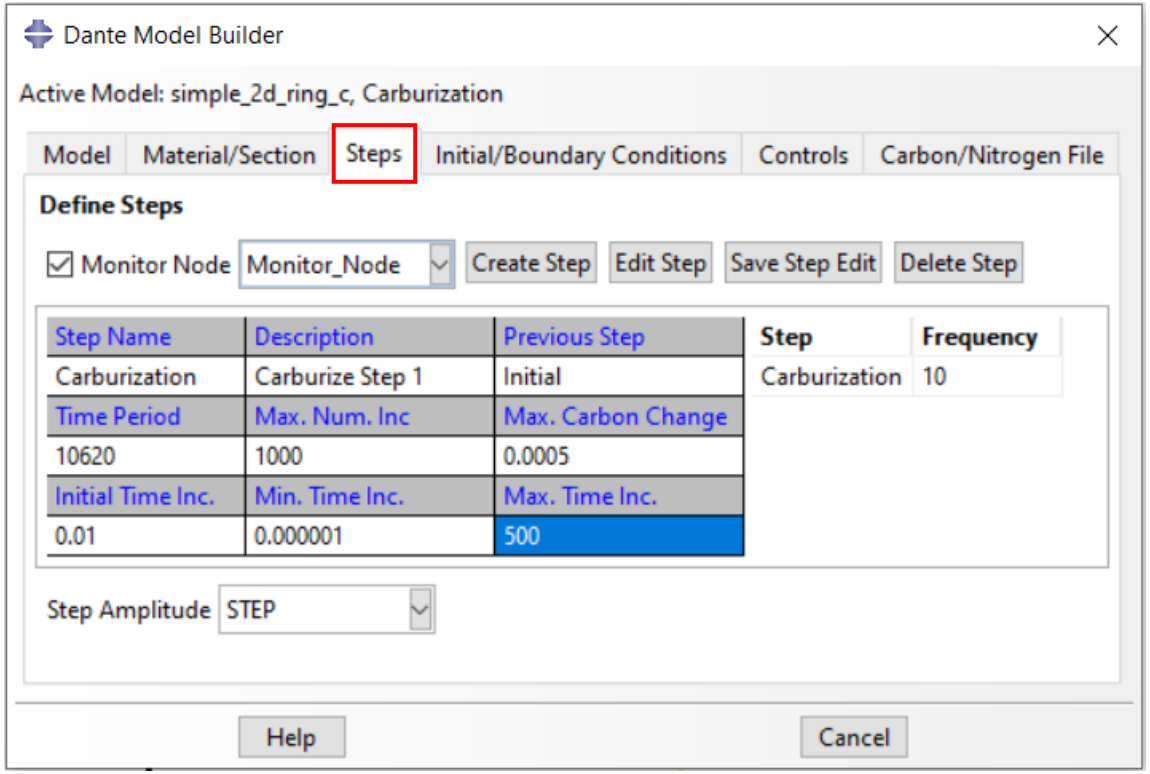
Step 9: Creating and Assigning Material





1. To create and assign material to the model, proceed to the ***Material/Section*** tab
2. Under the ***Part Element Set*** dropdown, select an element set
   - These are defined in the ***Part*** or ***Property*** Module and will automatically appear in the dropdown menu when created
   - A part element set was created in step 6 and should appear in the dropdown
3. Under the ***Material Name*** dropdown, select a Dante material
   - ***Initial Microstructure*** may be used to set the initial microstructure
   - **Alloy Composition** allows for slight variations to be made to the chemical composition of the selected alloy
4. Click ***Add Material Section*** and the material will be created, a section will be defined for the material, and a section assignment will also be created, assigning the section to the selected part set
   - A message box should appear if successful and can be dismissed

Step 10: Defining the Carburization Model Process Steps

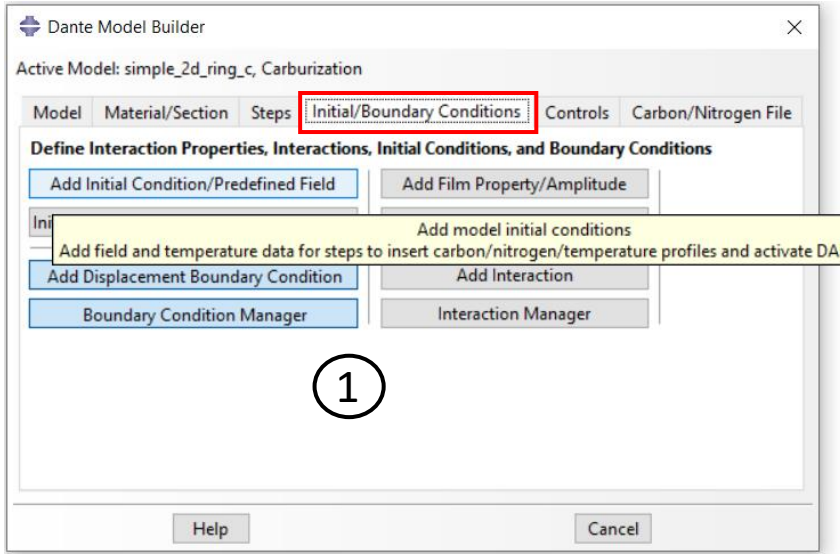


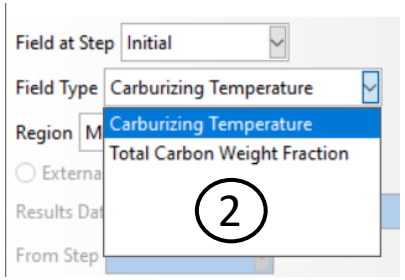
NOTE: Selecting a created step in the Step/Frequency table and clicking *Edit Step* will repopulate the step table with information from the selected step. This allows the user to edit and save step information with *Save Step Edit*

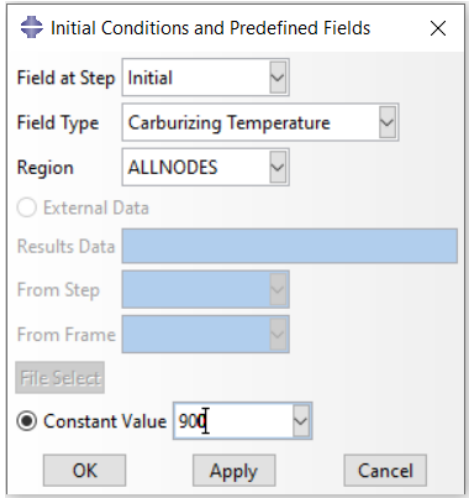
1. Select the ***Steps*** tab to create process steps
2. One step will be created, under the ***Step Name***, name the step 'Carburization', a ***Description*** can be provided, and the ***Previous Step*** should be set as shown in the figure
3. Under ***Time Period***, type 10620 seconds since this process step lasts 10620 seconds
4. The ***Max. Time. Inc.*** can be reduced to 500 seconds
5. Check the ***Monitor Node*** checkbox to create a monitor node for a given single node set defined as shown in step 6
6. Finally, click ***Create Step*** to create the process step, it should appear in the right-hand step/frequency table
   - Since this is a carburization model, an Abaqus heat transfer step is created for the model

Step 11: Creating Initial Conditions, Carburizing Temperature

- With the process steps defined, initial, boundary, and interaction conditions can be added



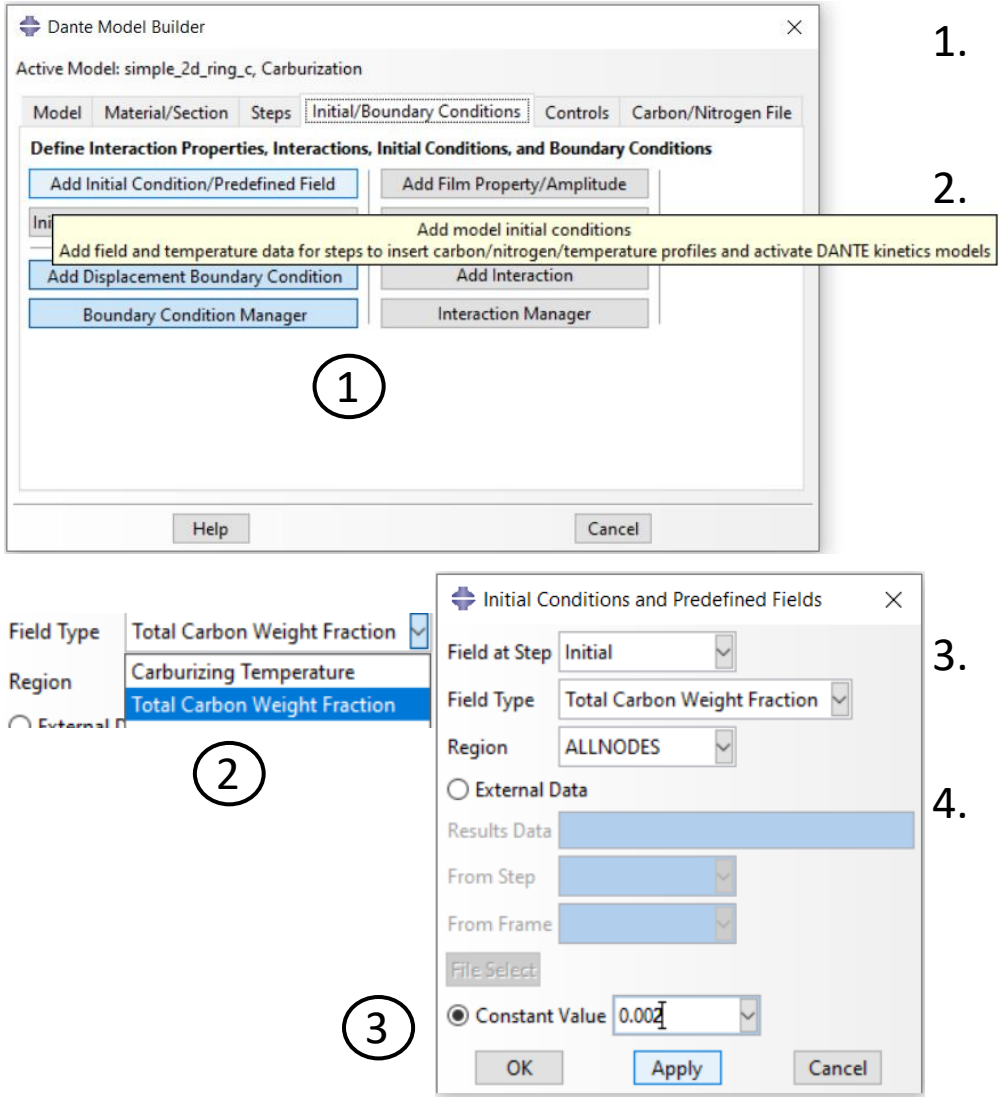




1. In the ***Initial/Boundary Conditions*** tab, click ***Add Initial Condition/Predefined Field*** to add initial conditions
2. There are two types of initial conditions that are needed for a DANTE carburization model: ***Carburizing Temperature*** and ***Total Carbon Weight Fraction***; both defined at the ***Initial*** step
3. The ***Carburizing Temperature*** is defined for all nodes, so a node set containing all nodes should be defined as in ***Step 6: Preliminary Sets, All Nodes***. This set will appear in the ***Region*** dropdown menu
4. Check the ***Constant Value*** checkbox to set a constant value uniform temperature in the carburized part and type in 900°C
5. Click ***Apply*** to create the initial condition and continue

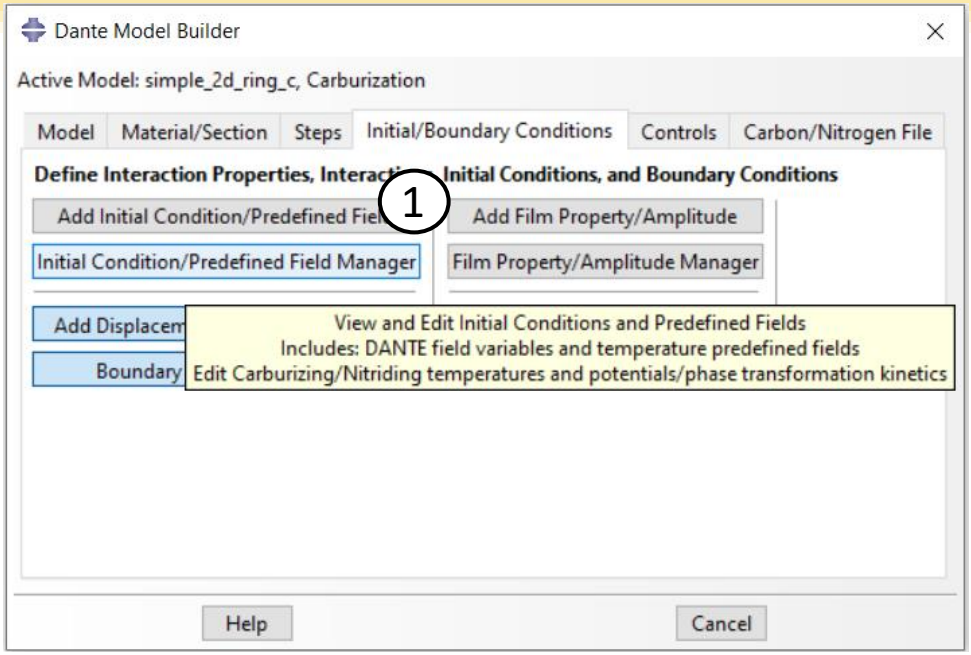
NOTE: To model multiple carburizing temperatures, multiple steps can be defined, and the predefined fields can be added to those steps

Step 12: Initial Conditions, Total Carbon Weight Fraction

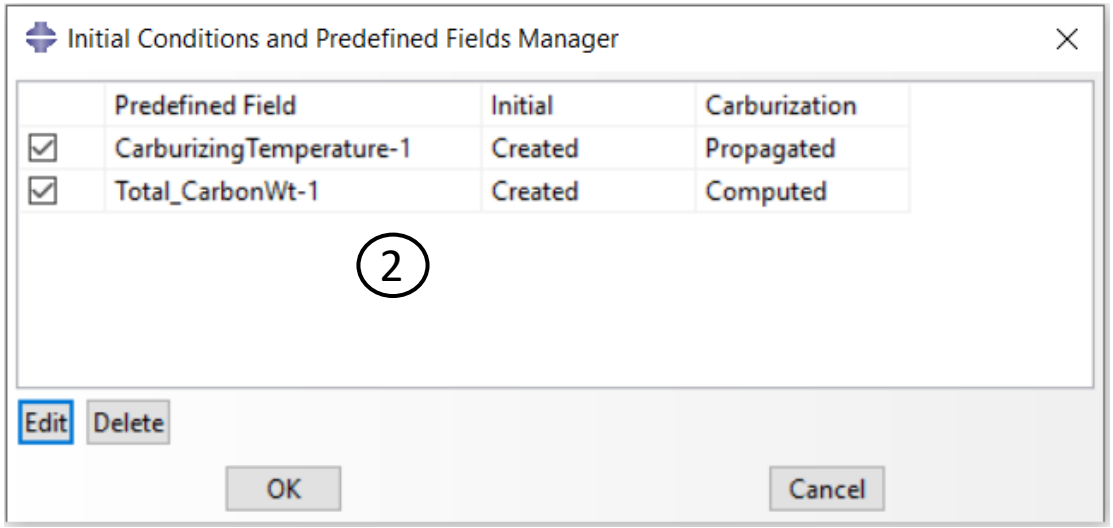


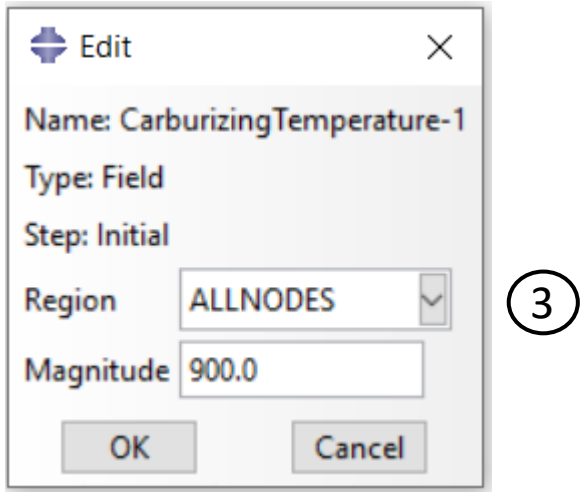
1. In the ***Initial/Boundary Conditions*** tab, click ***Add Initial Condition/Predefined Field*** to add initial conditions
2. The ***Total Carbon Weight Fraction*** is defined for all nodes, so a node set containing all nodes should be defined as in ***Step 6: Preliminary Sets, All Nodes***. This set will appear in the ***Region*** dropdown menu
   - Carbon weight fraction defined at the initial step is the initial distribution of carbon in the part. Either a constant uniform distribution of carbon can be specified OR a carbon profile can be provided through external data
3. In this case, check the ***Constant Value*** box and enter 0.002. This is the carbon content in 5120 steel, 0.2% carbon content or 0.002 carbon weight fraction
4. Click ***OK*** to create the initial condition and exit the dialog

Step 12b: Reviewing Initial Conditions and Predefined Fields

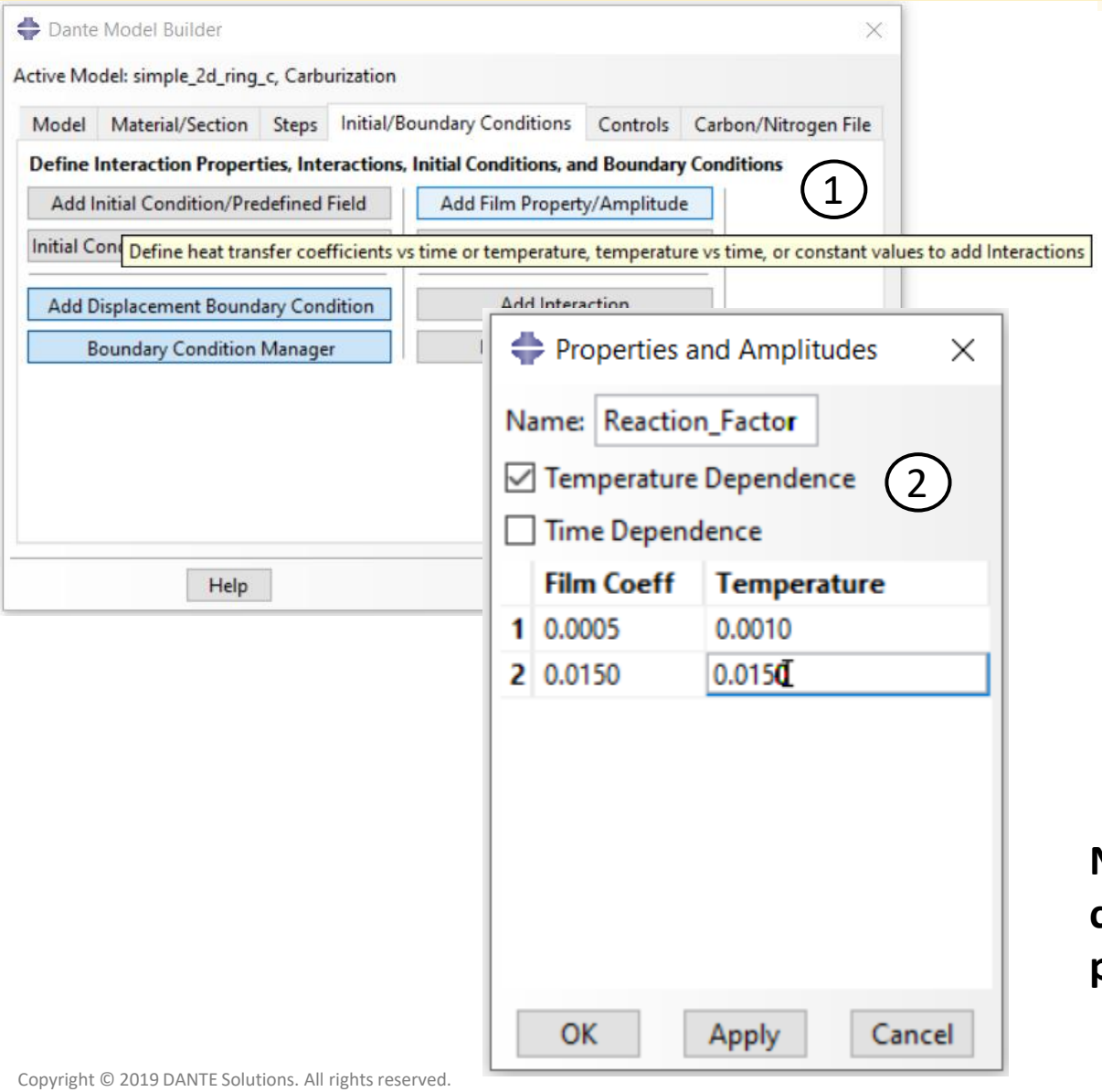


1. To review and edit initial conditions and predefined fields, click ***Initial Condition/Predefined Field Manager***
2. A table of the predefined fields will appear containing the status of each at each step. Click on any column of a row to highlight a predefined field
3. Click ***Edit*** to edit the predefined field or ***Delete*** to delete it
   - Use checkboxes to the left to suppress or unsuppress the predefined field, a check means the predefined field is unsuppressed





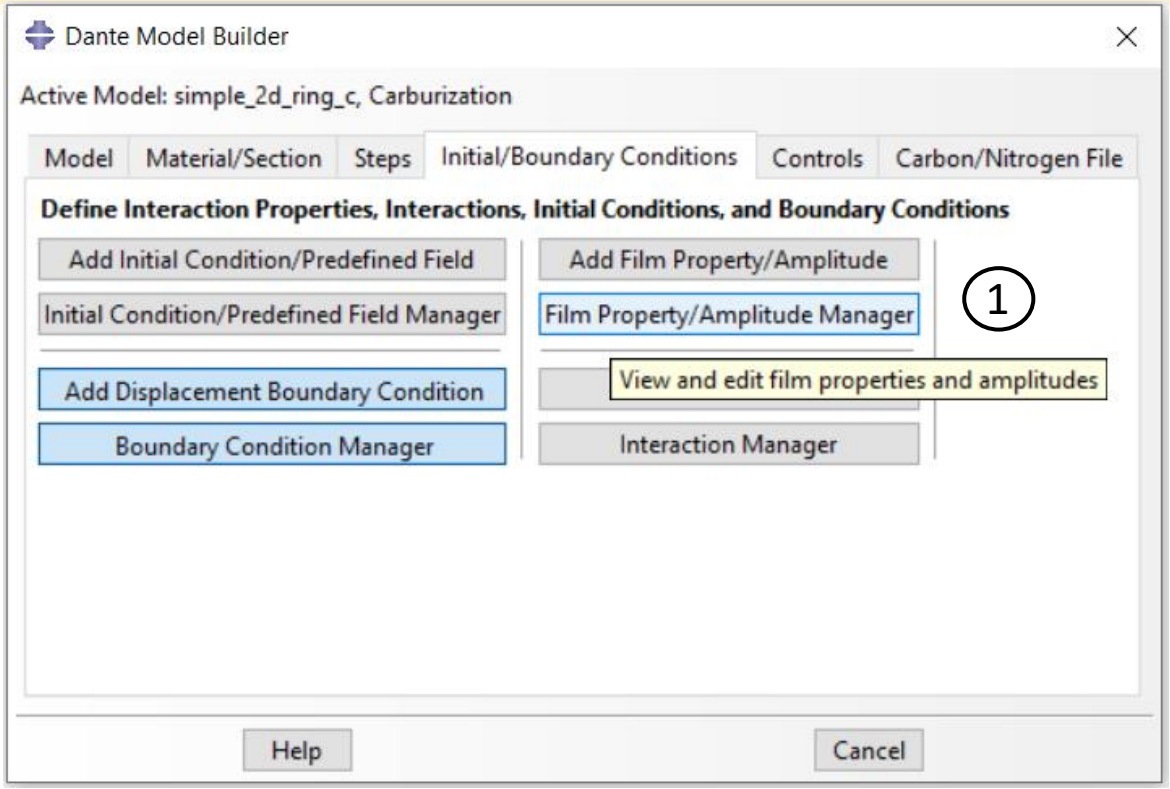
Step 13: Film Interaction Properties

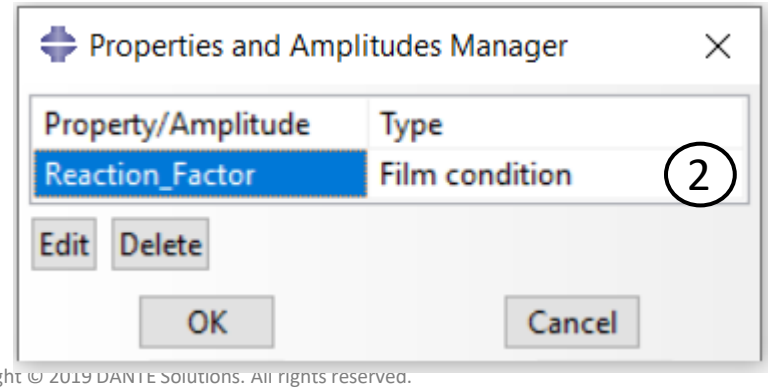


1. Heat transfer coefficients are needed to define the interactions of the environment and the surfaces of the part, so film interaction properties are created, click ***Add Film Property/Amplitude***
2. Give the name ‘Reaction_Factor’
3. Film coefficients as a function of temperature are defined which represents the rate at which carbon is deposited on the surface of the part as a function of surface carbon, so check the ***Temperature Dependence*** checkbox
4. Enter in the film coefficient vs temperature data: (0.0005,0.0010), (0.0150,0.0150)
5. Click ***OK*** to create the property and close the dialog

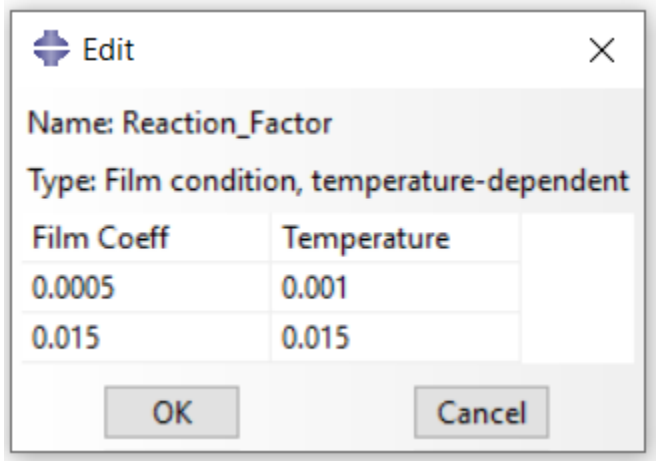
NOTE: Recall the temperatures in the data are carbon content as carbon diffusion models are identical in form to heat transfer problems, so heat transfer steps are used

Step 13b: Reviewing Film Interaction Properties

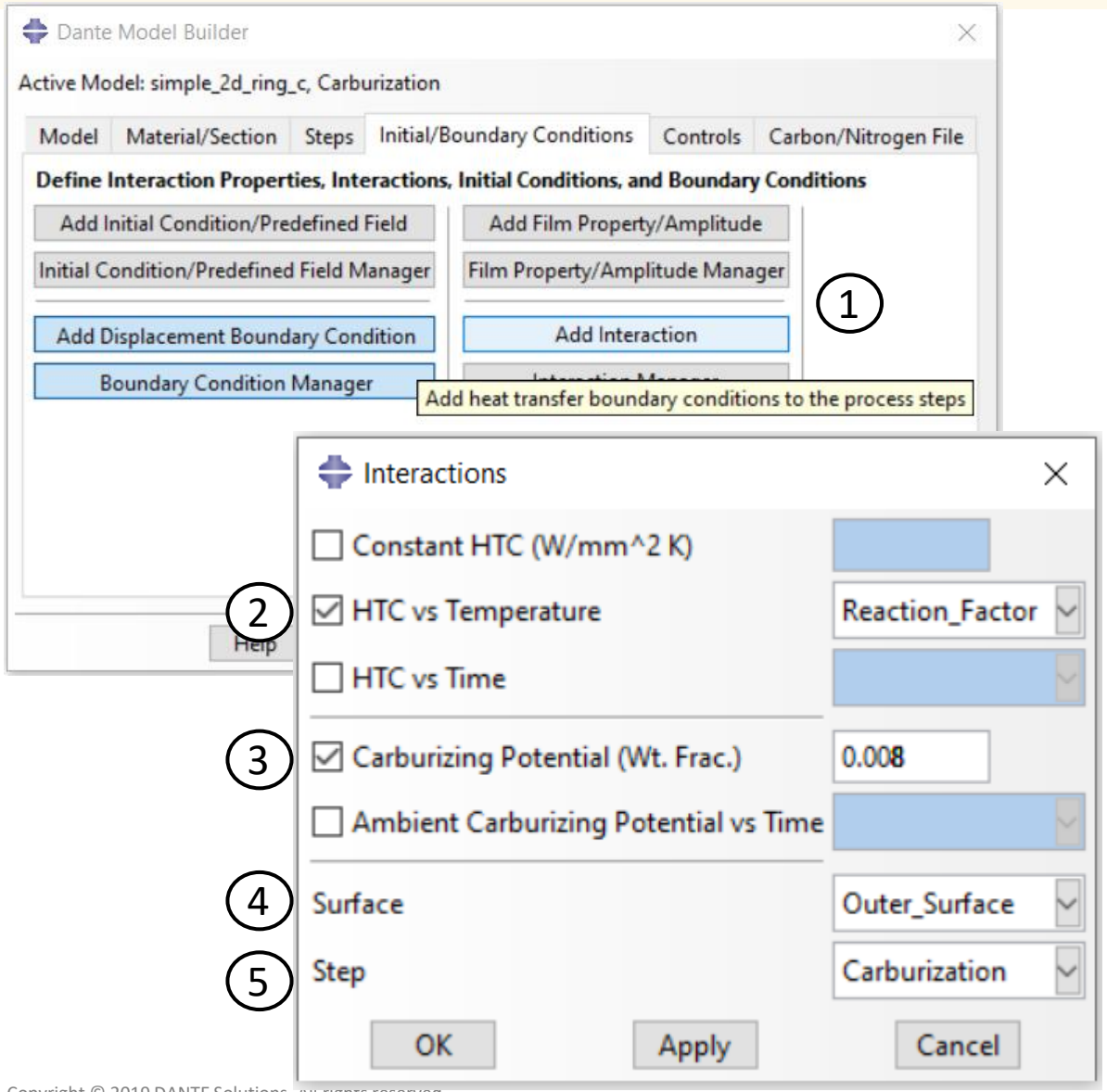




1. To review and edit interaction properties or amplitudes, click ***Film Property/Amplitude Manager***
2. A table of the interaction properties and amplitudes will appear containing the type. Click on any column of a row to highlight a property or amplitude
3. Click ***Edit*** to edit the item or ***Delete*** to delete it
    - For the film property created, changes can be made to the film coefficients vs temperature if needed

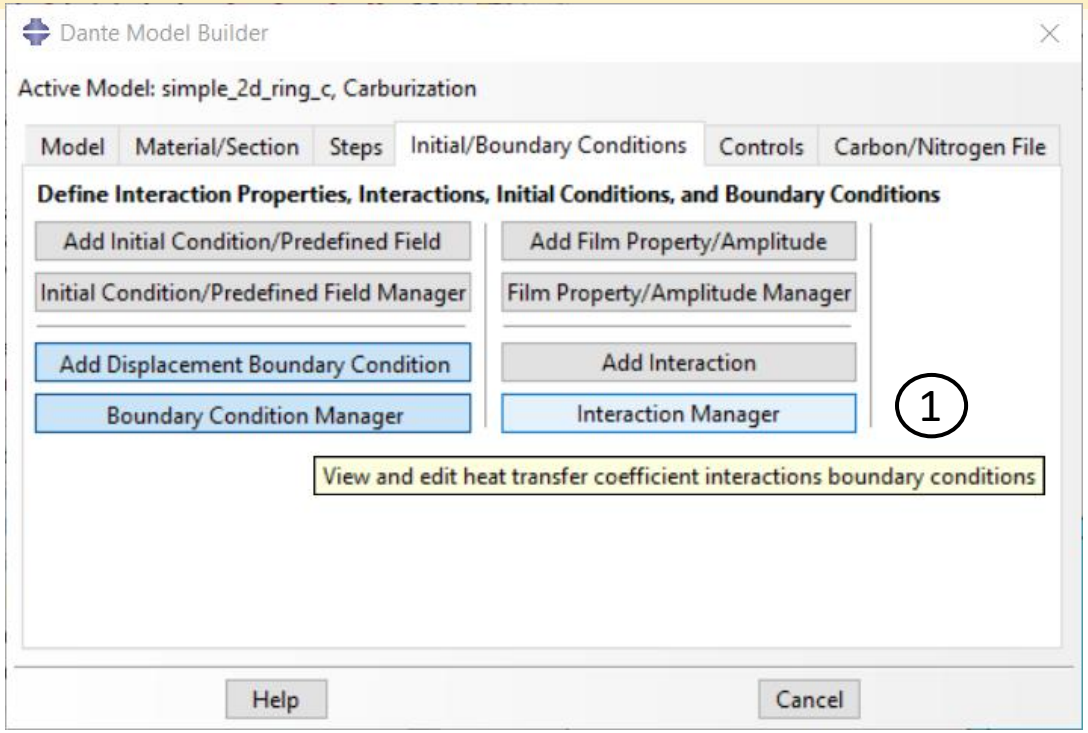


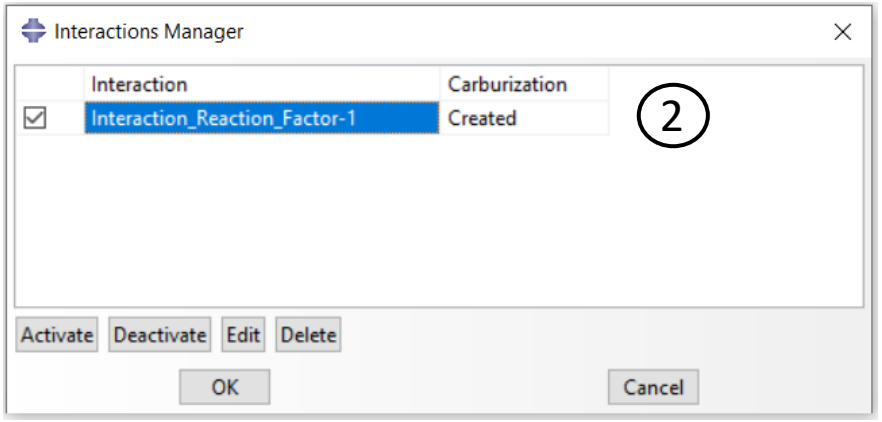
Step 14: Adding Interactions



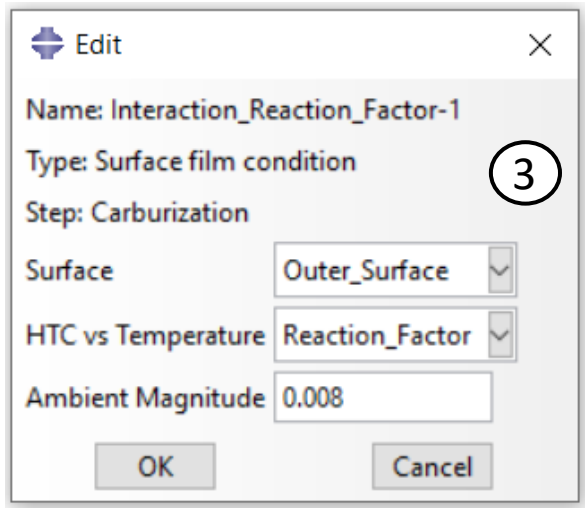
1. Finally the heat transfer interactions can be added to the model based on the film property added in the previous step, click ***Add Interaction***
2. The ***HTC vs Temperature*** option should be checked
3. Check ***Carburizing Potential (Wt. Frac.)*** and enter in 0.008
   - This is the target surface carbon content weight fraction. That is, the 5120 material is being carburized to 0.8% carbon content
4. All surfaces of the ring are carburized, so all surfaces in this 2D case should be selected in a surface object. This surface is the 'Outer_Surface' defined in ***Step 6: Preliminary Surfaces***
5. Select the first step, Carburization. Thus, the part will be carburized to a 0.8% carbon content in the first step, subject to furnace conditions defined by the 'Reaction_Factor' film interaction property
6. Click ***OK*** to apply the property and close the dialog

Step 14b: Reviewing Interactions

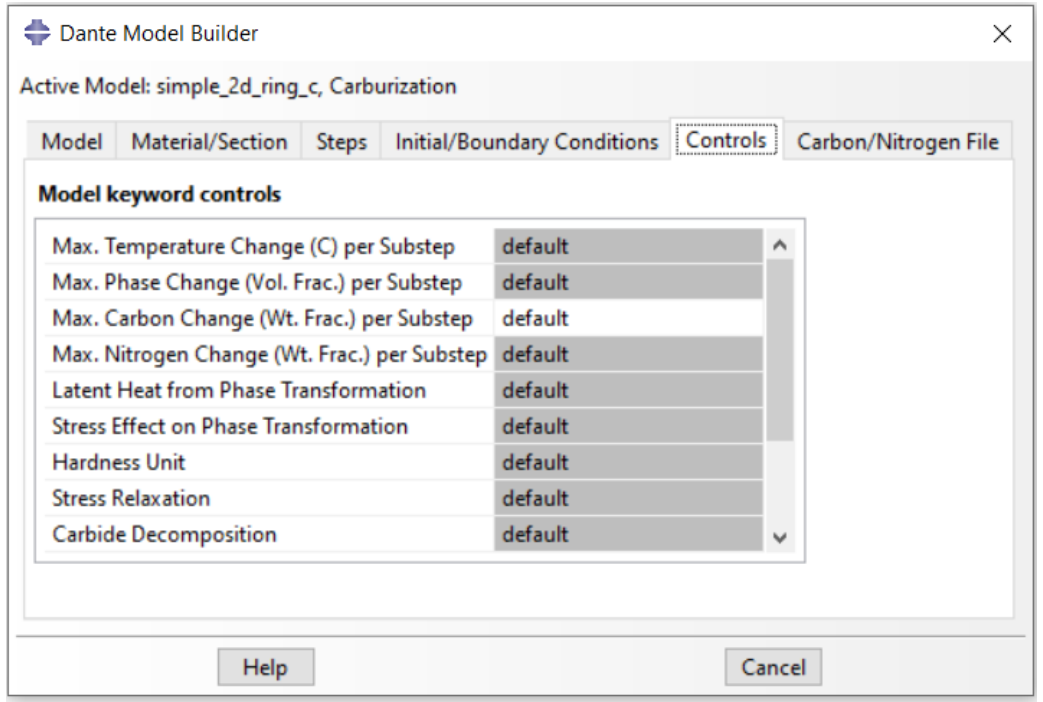




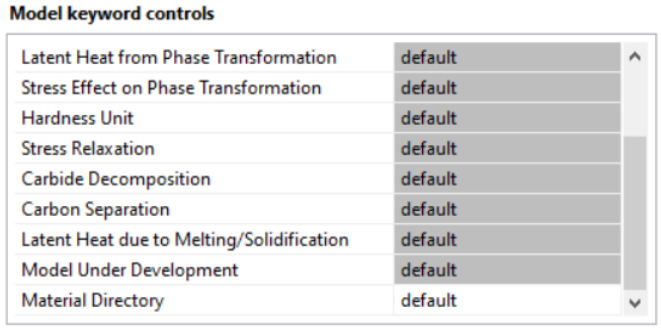
1. To review and edit interactions, click ***Interaction Manager***
2. A table of the interactions will appear containing the status in each step. Click on any column of a row to highlight an interaction
   - The left-hand column check boxes can be used to suppress interactions, ***Activate*** and ***Deactivate*** can be used to turn the interaction on or off in a certain step
3. Click ***Edit*** to edit the item or ***Delete*** to delete it
   - The selected surface, film property, and ambient magnitude (in this case carburizing potential) can be edited from here



Step 15: Configuring Carbon Model DANTE Controls

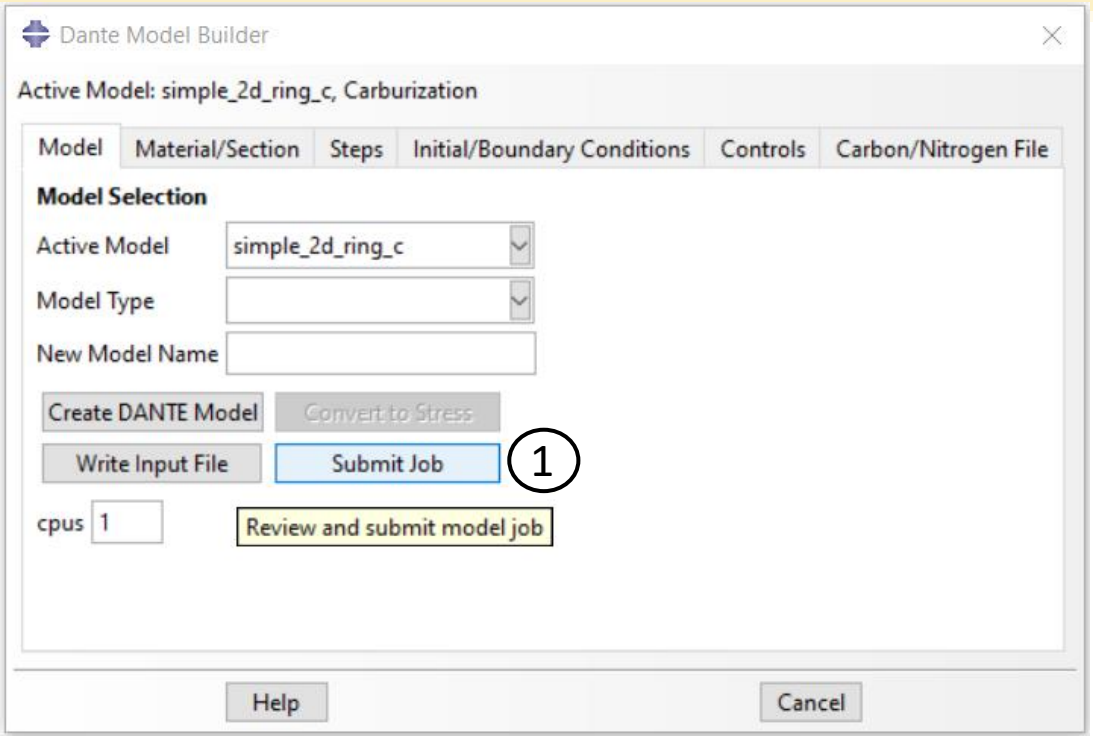


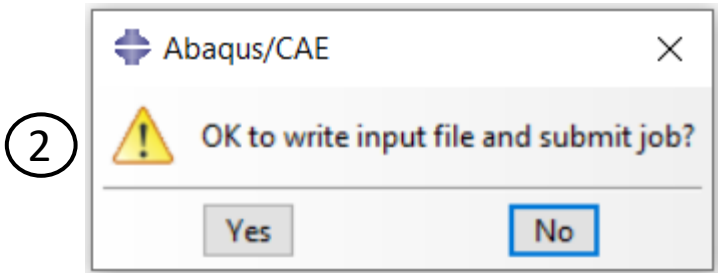
- Carburization models allow for control of the maximum carbon change per substep and the material directory
- The default values will suffice
- Grayed out rows cannot be edited in the respective model type



- This concludes the set up of the carburization model
- The next step is to submit the job, so be sure to save the model beforehand

Step 16: Creating and Submitting the Carburization Job





NOTE: At this point, the Plug-In may be exited by clicking cancel or 'x' out in the corner

- At this point, the carburization model is complete, and a job can be created. Note that an input file can be written without submitting the job. The input file will get written regardless and, in both cases, checks are applied to verify that the proper DANTE model parameters have been specified for the respective model type

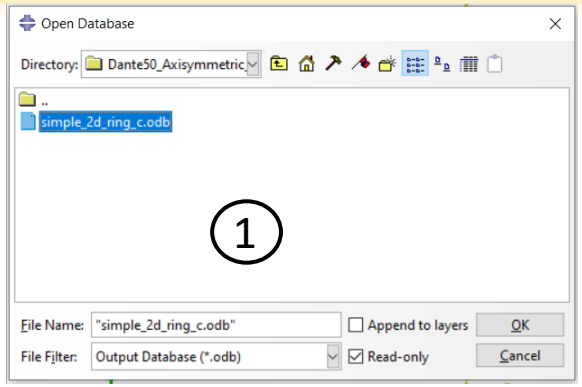
1. A new model name can be given for the job name if desired in the ***New Model Name*** box, skip this here to use the original model name 'simple_2d_ring_c' as the job name

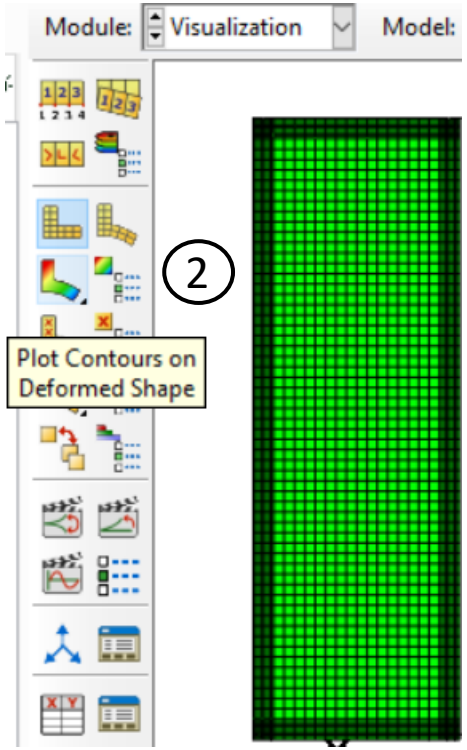
2. Click ***Submit Job*** and the following dialog box will appear asking for confirmation to submit the job, click ***Yes***

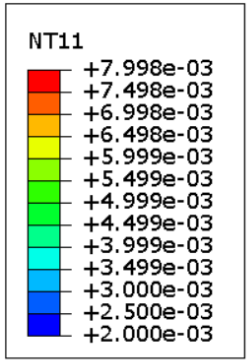
    - This will run a series of checks in the model to verify it, if something is found to be problematic, an error message box should appear. Otherwise, the CAE should switch over to the ***Job*** module and text confirming the run steps should appear in the CAE console. This indicates if the model was input successfully:

```
History output is not requested in the following steps:
Carburization
Job simple_2d_ring_c: Analysis Input File Processor completed successfully.
Job simple_2d_ring_c: Abaqus/Standard completed successfully.
Job simple_2d_ring_c completed successfully.
```

Step 17: Carburization Model Post-Processing



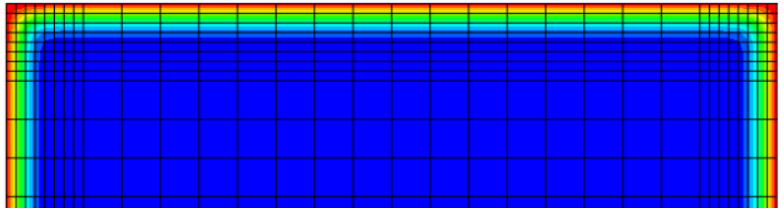




1. With the model completed or even in simulation, the results file, or Abaqus output database, can be opened from ***File → Open***, select *.odb in ***File Filter*** and the odb file 'simple_2d_ring_c.odb' and click ***OK***

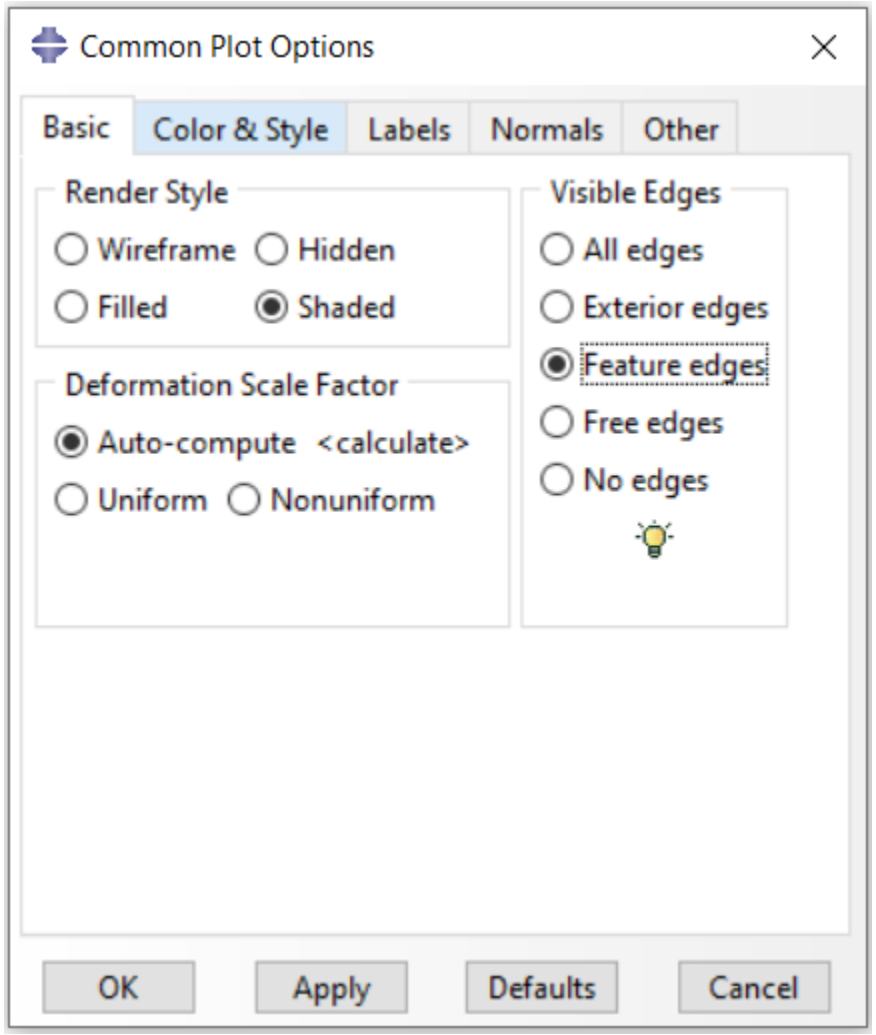
2. By opening the odb file, the CAE will switch to the ***Visualization*** tab, click on the ***Plot Contours on Deformed Shape*** button to display result contours for the temperature at the final time increment of the model

   - Note that the temperature variable NT11 is displayed and that it represents the carbon content weight fraction due to the use of heat transfer steps in the Abaqus model
   - Thus, this contour image shows the carbon distribution along a cross-section of a ring slice that is carburized from 0.2% Carbon to 0.8% Carbon content by weight

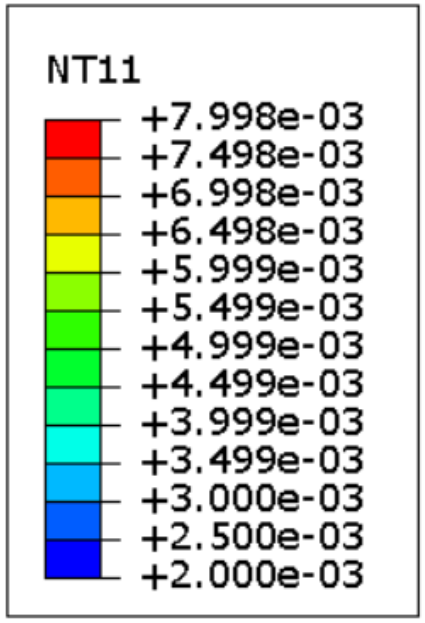


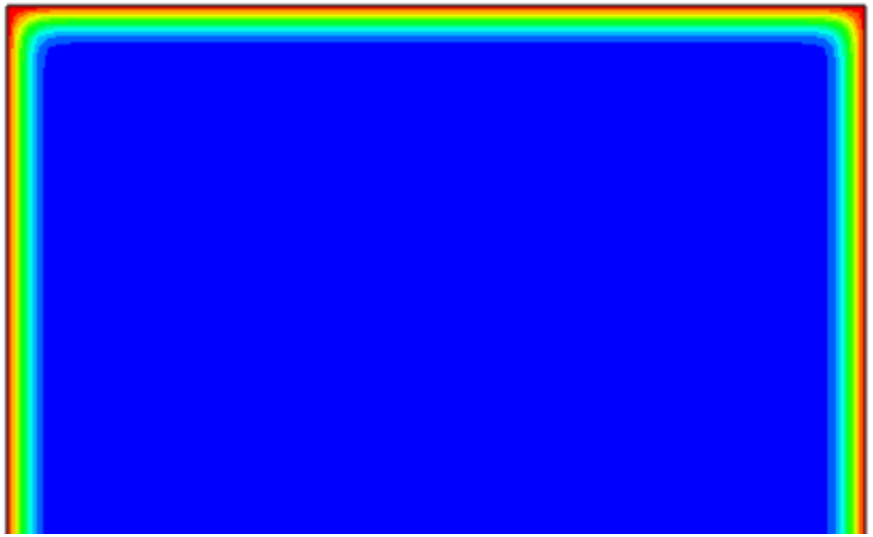
Step 17b: Carburization Model Post-Processing

Options → Common



- By setting the ***Visible edges*** to ***Feature edges*** through the ***Options → Common*** settings, the contours will be shown without the meshing
- This can be helpful to better visualize results

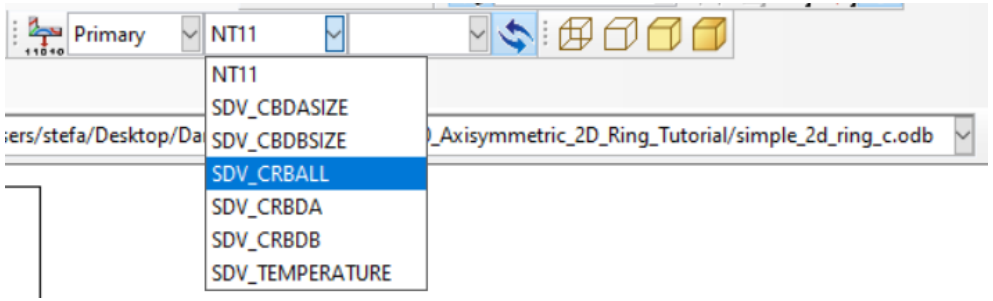


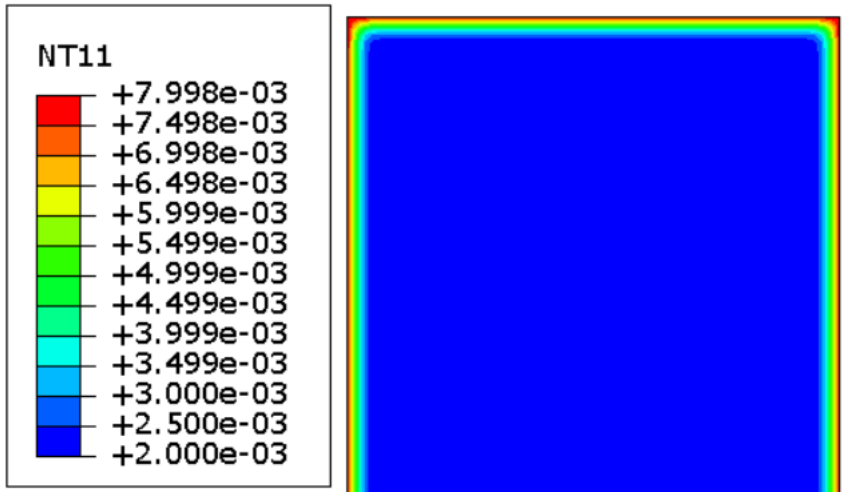


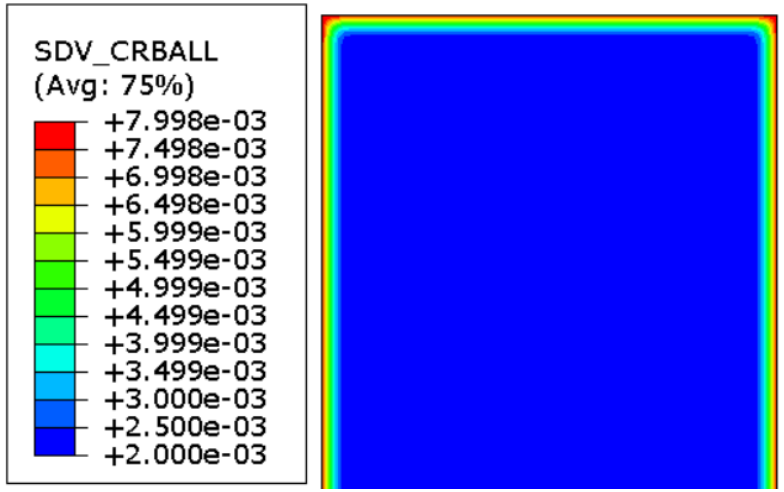
Step 18: Visualizing Carburization Solution Dependent Variables

- The other results may be visualized by selecting solution dependent variable results from the dropdown menu in the Visualization tab
- These variables were automatically assigned to the model input file through the Dante Model Builder Plug-In based on the assigned DANTE model type
- For this material and model, ***SDV_CRBALL*** shows the total Carbon weight fraction and should be similar to NT11 with slight differences caused by numerical precision

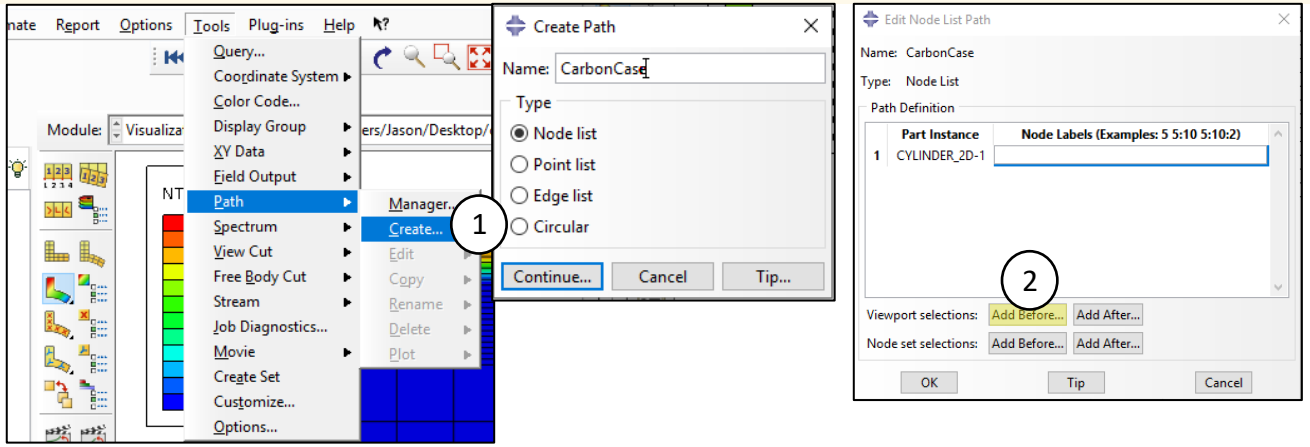
NOTE: For further information on post-processing of these results, please refer to the tutorial:
Dante5_0_Tutorial_2D_Axisymmetric_Ring_Slice_CAE

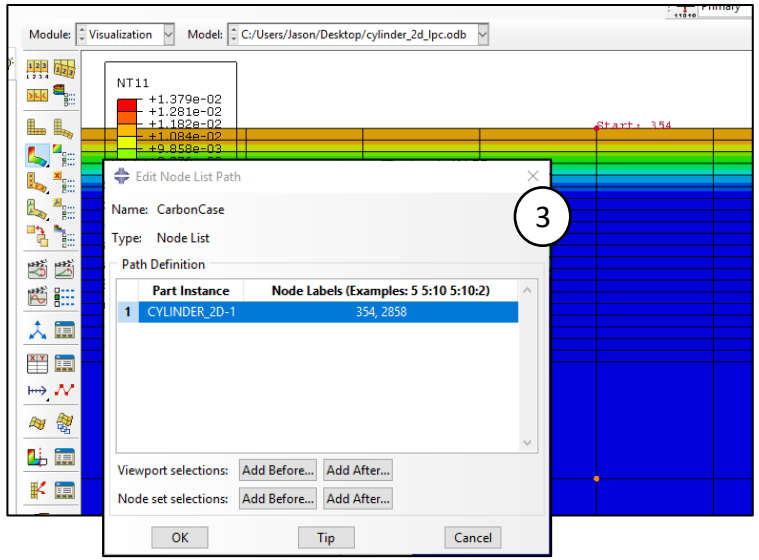


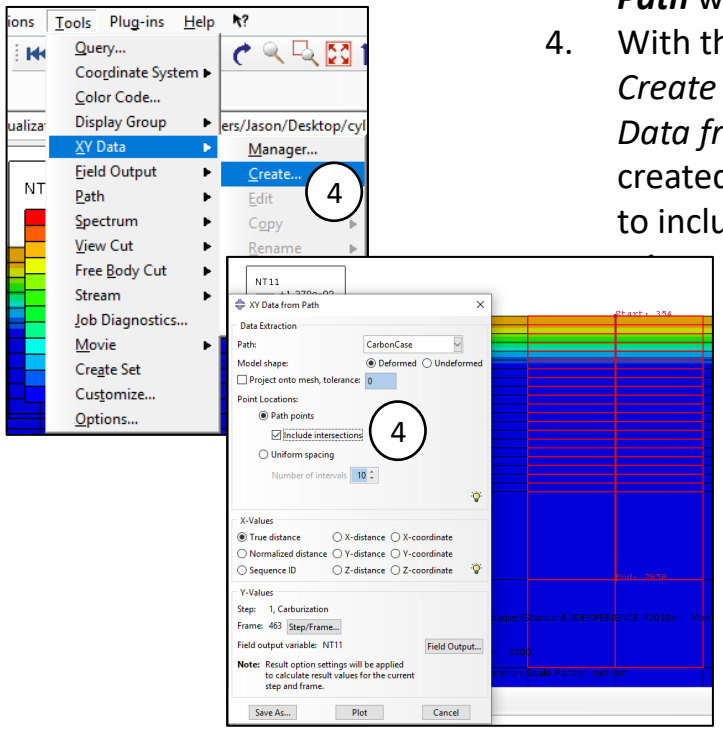




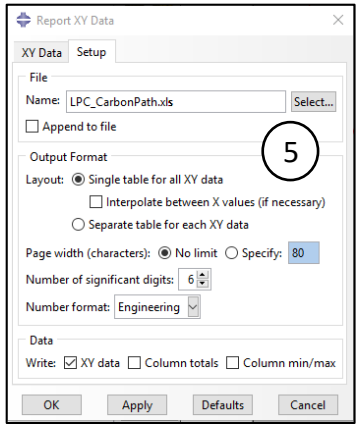
Step 18b: Set up a Path Plot





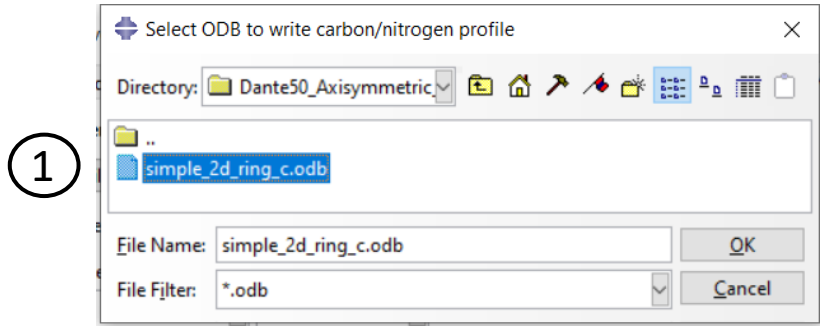


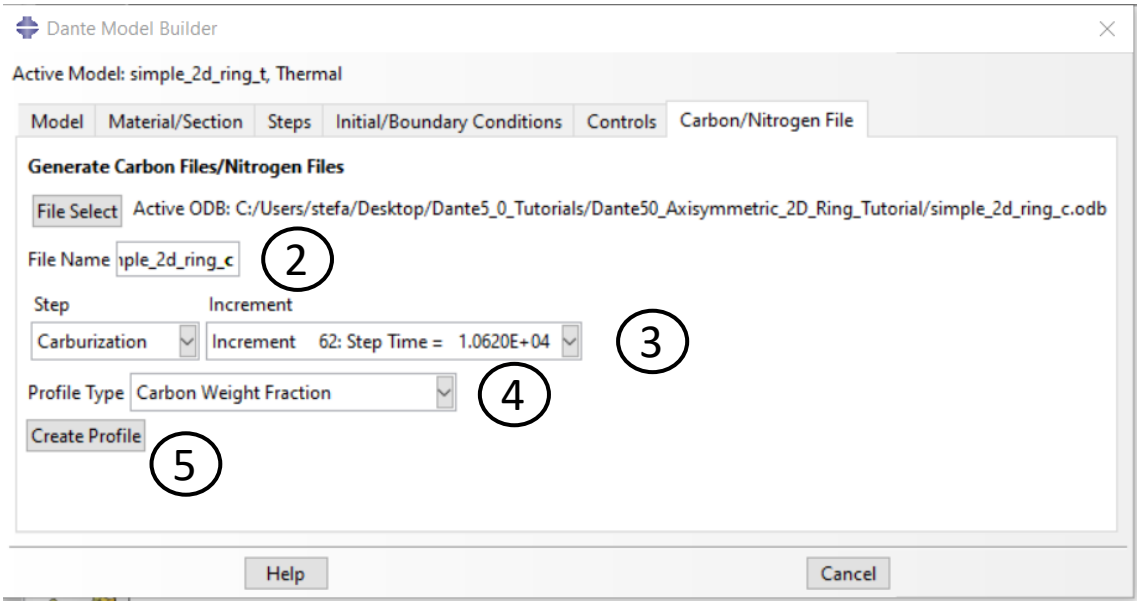
1. A path plot is desirable for carbon in order to easily see carbon case depths. While in the ***Visualization*** module, go to ***Tools → Path → Create*** to open the ***Create Path*** dialog window. Name the path and select ***Node list*** and click ***Continue*** to select the nodes to make the path.
2. In the ***Edit Node List Path*** popup window click on ***Add Before…*** in the viewport selections section
3. Click on a node on the surface of the part and follow the mesh line to the core of the part and select a node to use as the end of the path plot. Middle click or select done to set the nodes in the ***Edit Node List Path*** window and click ***OK*** to continue.
4. With the path defined, go to ***Tools-> XY Data-> Create*** to open the *Create XY Data* window. Choose ***Path*** and click ***Continue***. In the *XY Data from Path* window that opens, ensure that the path that was created is selected and check the box that states *Include intersections* to include every node along the path. Finally, click plot to show the Carbon vs. Depth plot.
5. From the Report menu, select ***XY…*** to open the ***Report XY Data*** window. In the XY Data tab select the data in the ***current viewport***, and under the ***Setup*** tab rename the file what you wish with a .xls extension to save the carbon path data. Click ***OK*** or ***Apply*** to write the carbon data to the Exel file.



Step 19: Generating the Carbon Distribution File

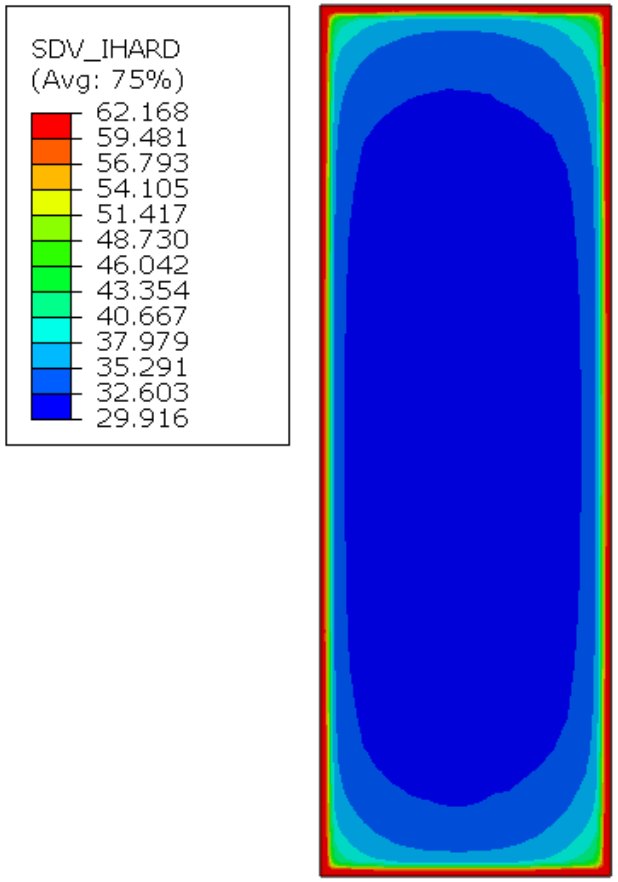
- The Carbon distribution in the component can be written out into an external file and used in the thermal and stress models

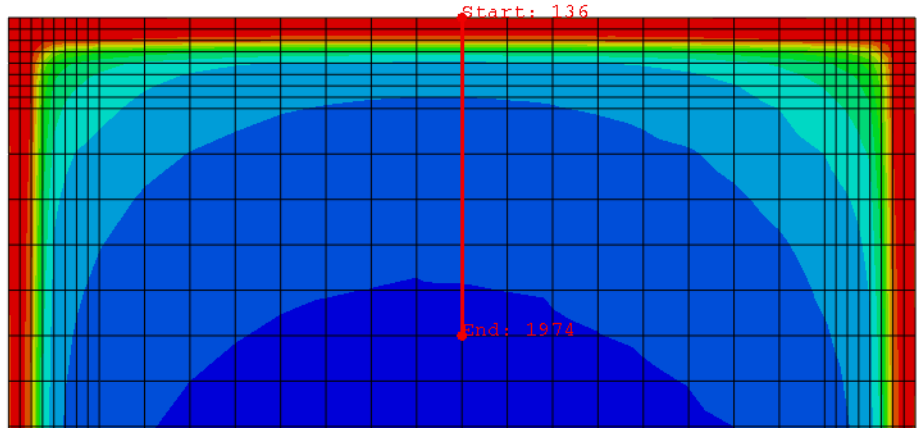


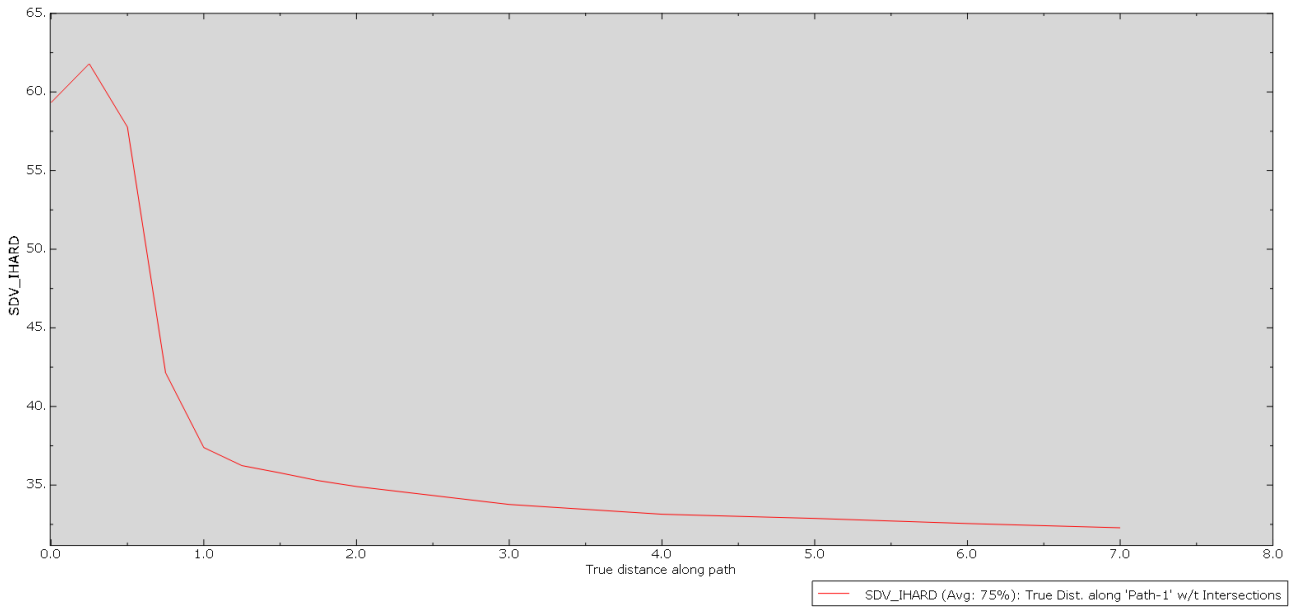


1. To write out the carbon profile of the part from the end of the carburization process, go to the ***Carbon/Nitrogen File*** tab in Dante Model Builder and click ***File Select*** and open the output database '**simple_2d_ring_c.odb**', the file name should appear next to ***Active ODB:***
2. Give a name for the carbon profile to be generated, 'simple_2d_ring_c', here it will be given a '.cbn' extension automatically
3. The ***Step*** and ***Increment*** dropdown menus will be populated with the available steps and step increments from the output database, select the last step and last increment in the list to write out the ***final*** carbon weight fraction
4. For the ***Profile Type***, ***Carbon Weight Fraction*** should be selected
5. Click ***Create Profile*** to write out the carbon file, a file with the name and extension: 'simple_2d_ring_c.cbn' will be written out to the current working directory

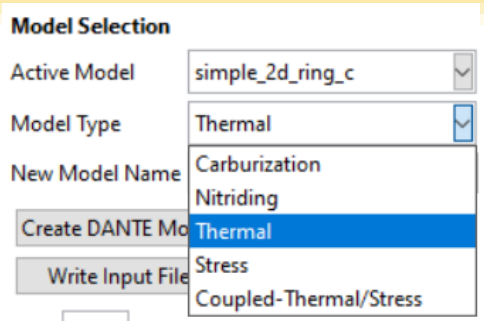
Thermal Model



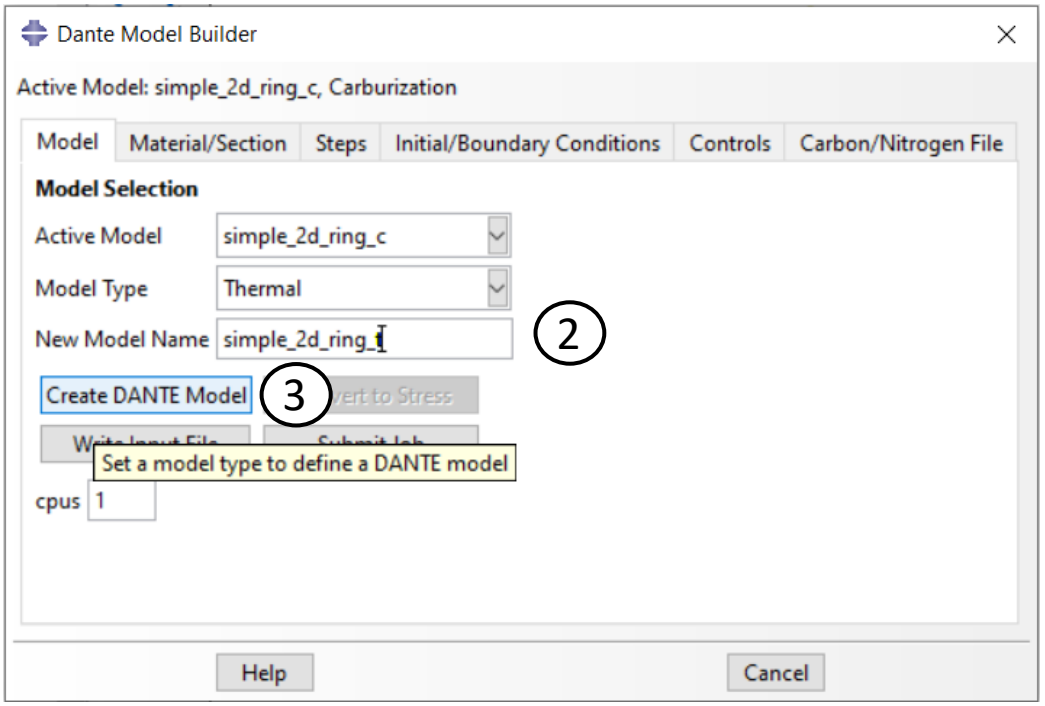




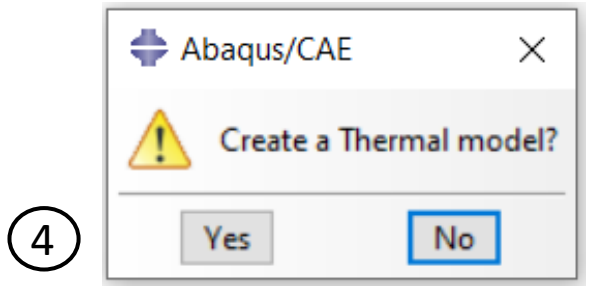
Thermal Model



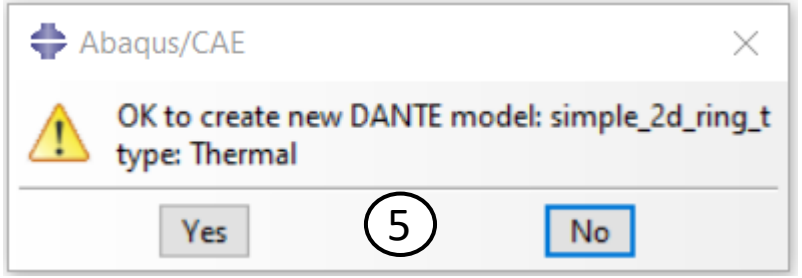
1. To create a thermal model, a new model can be added under the model tree, or in the Plug-In, select ***Thermal*** for ***Model Type*** in the ***Model*** tab, with the carbon model as the ***Active Model***
2. Give a ***New Model Name***, here 'simple_2d_ring_t'

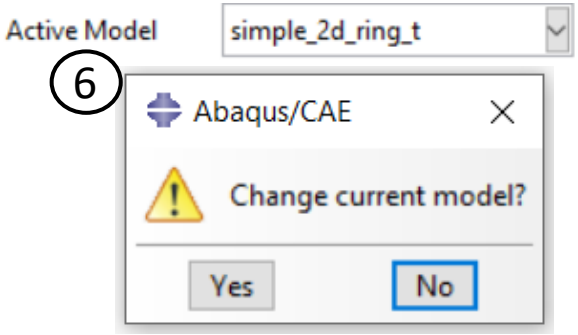


3. Click ***Create DANTE Model***
4. Click ***Yes*** to create the model



5. Since a model name and type are given, the active model will be ***copied*** and a new model should appear in the model tree if ***Yes*** is clicked in the next dialog box asking to create the model
6. Switch to the newly created Thermal model in the ***Active Model***

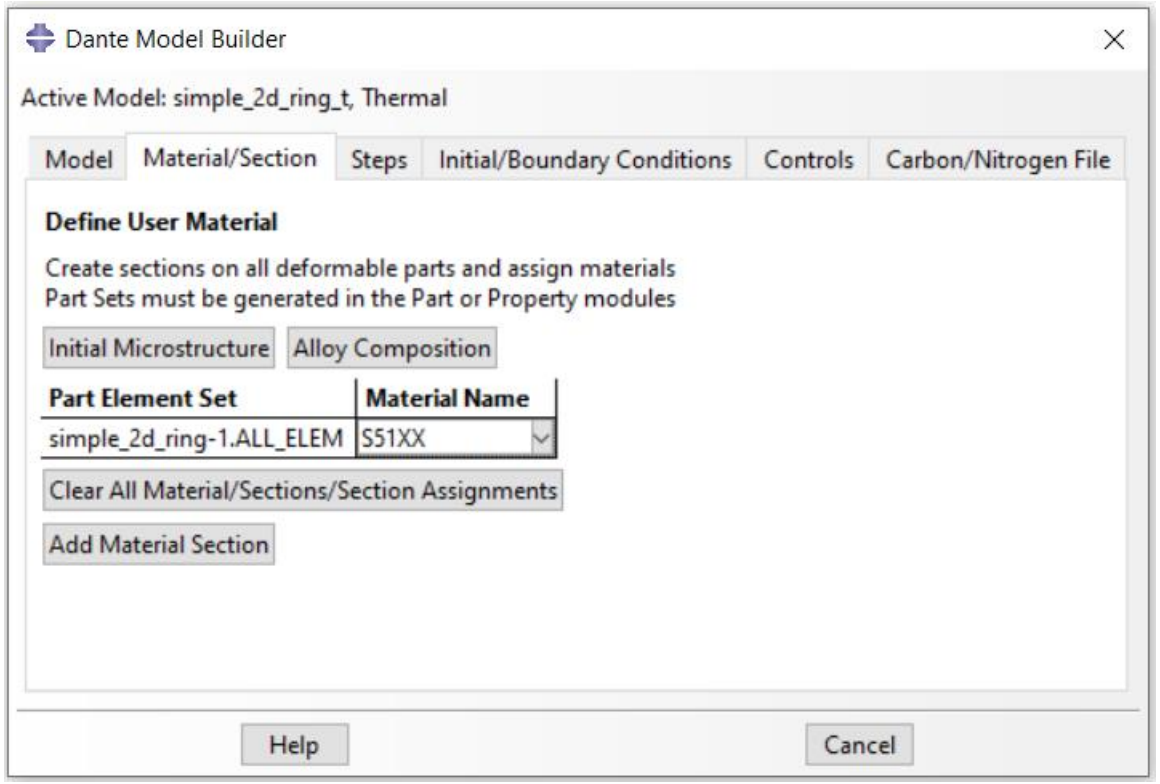




NOTE: Be sure to save the model at this point

Step 1: Creating and Assigning Material

- Materials and sections must be defined and assigned to the proper regions in the geometry. This is done from the ***Material/Section tab*** in the same way as in ***Step 9: Creating and Assigning Material*** and the material and element set should be the same



Step 2: Defining the Thermal Model Process Steps

- The thermal model steps are defined like the carburization model steps from step 10, as follows:

1

Step Name	Description	Previous Step
Heat-Up	Furnace Heat Up	
Time Period	Max. Num. Inc	Max. Temp. Change
1000	100	30
Initial Time Inc.	Min. Time Inc.	Max. Time Inc.
1	0.000001	1000

Step	Frequency
Heat-Up	10

2

Step Name	Description	Previous Step
Carburization	Carburization Process	Heat-Up
Time Period	Max. Num. Inc	Max. Temp. Change
1	500	30.0
Initial Time Inc.	Min. Time Inc.	Max. Time Inc.
1.0	1e-06	1

Step Amplitude: RAMP

Step	Frequency
Heat-Up	10
Carburization	10

3

Step Name	Description	Previous Step
Air-Transfer	Transfer from Furnace	Carburization
Time Period	Max. Num. Inc	Max. Temp. Change
12	1000	30
Initial Time Inc.	Min. Time Inc.	Max. Time Inc.
0.01	0.000001	12

Step	Frequency
Heat-Up	10
Carburization	10
Air-Transfer	10

***NOTE*: All steps should have the *Step Amplitude* set to *Step EXCEPT* for the *Carburization* step where the Carbon profile will be mapped into the thermal model. Set Step Amplitude to RAMP when mapping in Carbon profiles**

4

Step Name	Description	Previous Step
Oil-Quench	Oil Quenching Process	Air-Transfer
Time Period	Max. Num. Inc	Max. Temp. Change
1000	1000	30
Initial Time Inc.	Min. Time Inc.	Max. Time Inc.
0.01	0.000001	50

Step	Frequency
Heat-Up	10
Carburization	10
Air-Transfer	10
Oil-Quench	10

5

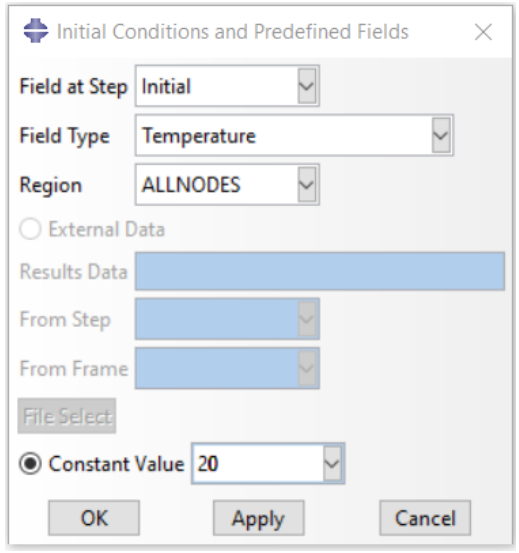
Step Name	Description	Previous Step
Air-Cool	Air Cool to Room Tem	Oil-Quench
Time Period	Max. Num. Inc	Max. Temp. Change
1500	1000	30
Initial Time Inc.	Min. Time Inc.	Max. Time Inc.
1	0.000001	100

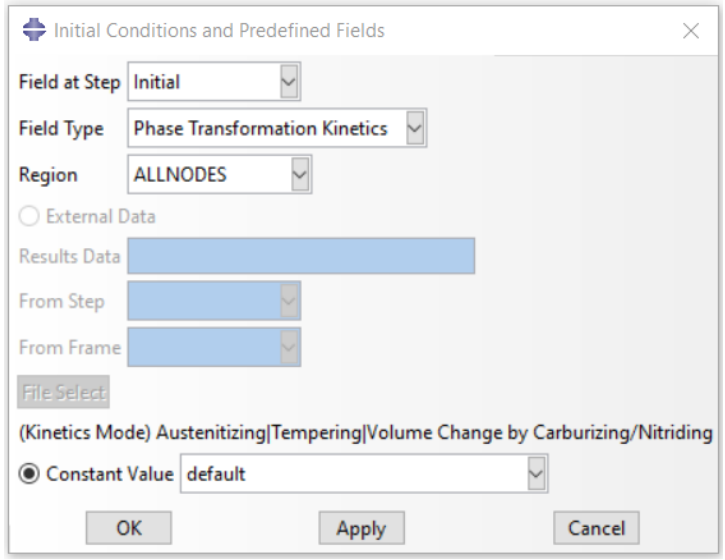
Step	Frequency
Heat-Up	10
Carburization	10
Air-Transfer	10
Oil-Quench	10
Air-Cool	10

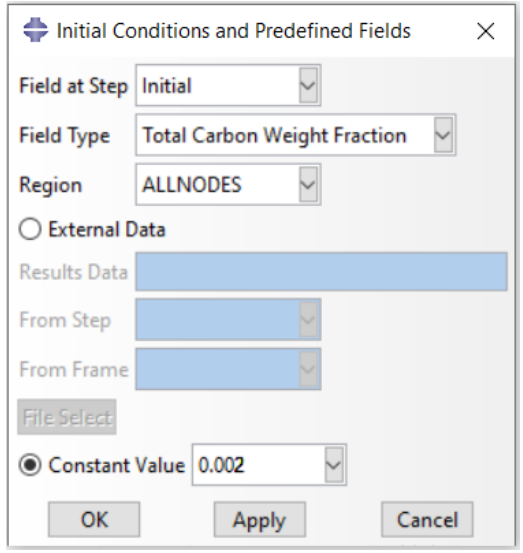
- Be sure to check the ***Monitor Node*** check box and select the monitor node set. Once completed, ***save*** the CAE model

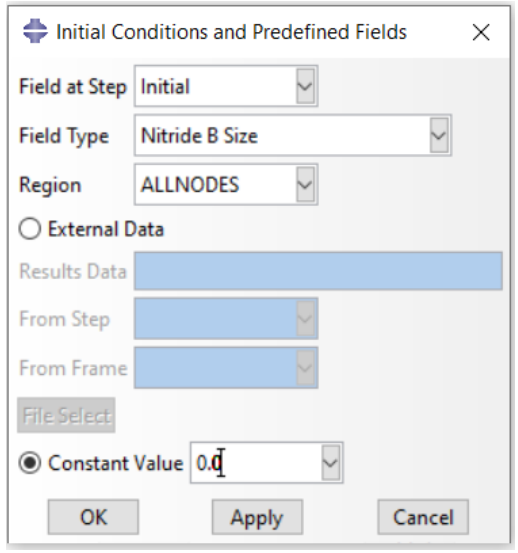
Step 3: Creating Initial Conditions

- Thermal models require at least 4 types of initial conditions: Temperature, Phase Transformation Kinetics, Total Carbon Weight Fraction, and Nitride B Size Factor (which is added automatically so long as the others are defined properly)

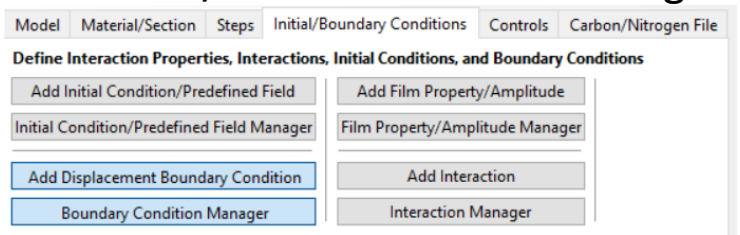


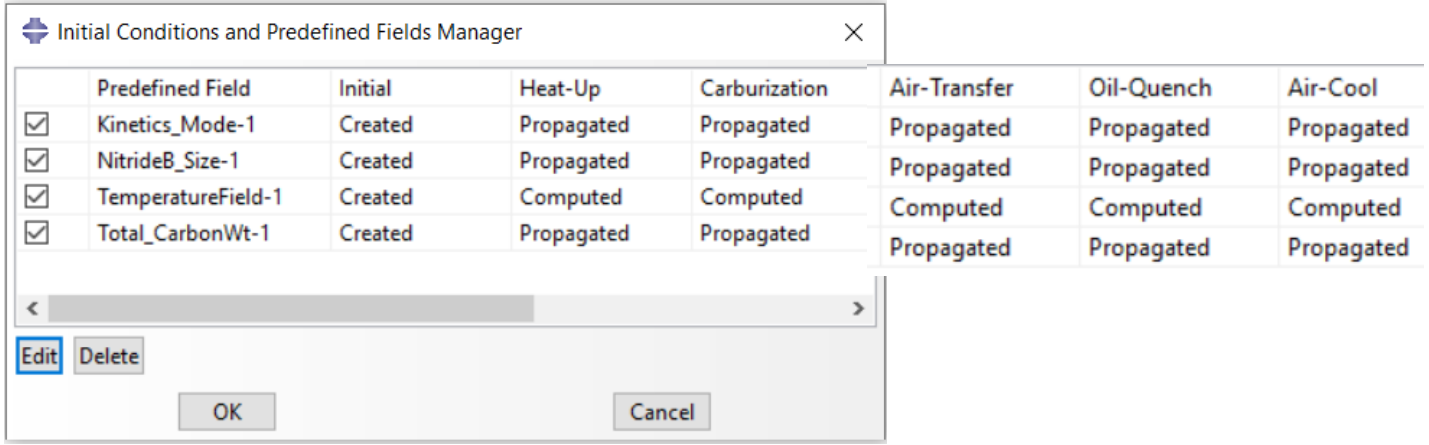






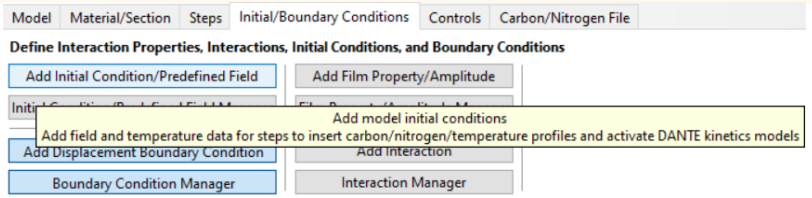
- These initial conditions can be reviewed in the Initial Condition/Predefined Field Manager

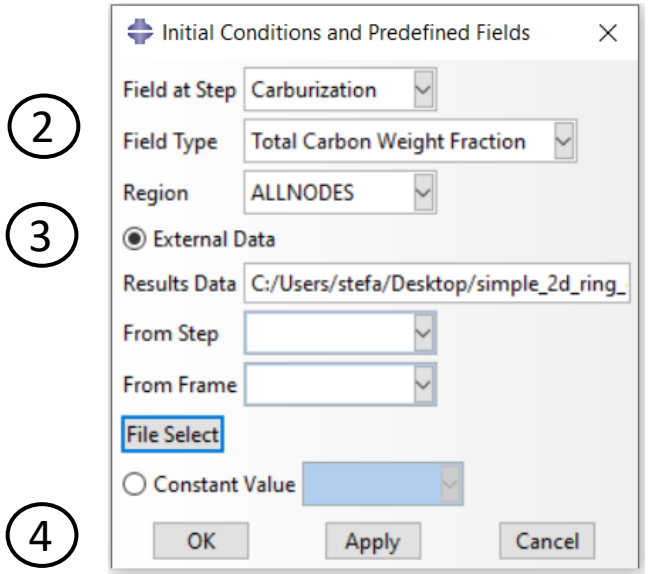


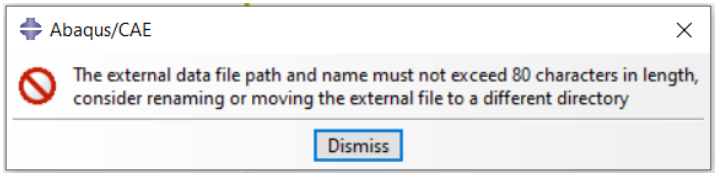


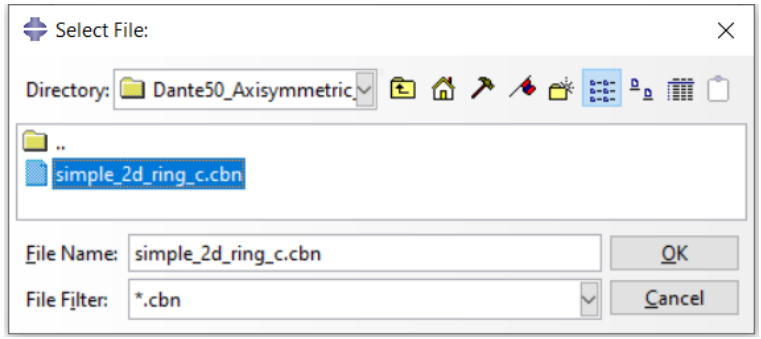
Step 4: Predefined Field to Map In Carbon Profile

1. To map in the carbon profile, the external .cbn file created previously will be read in at the Carburization step (where it will be gradually ramped into the model)
2. Carbon will be read in as a ***Total Carbon Weight Fraction*** field in the set of all nodes, defined in **ALLNODES**
3. Check ***External Data*** and ***File Select*** the carbon profile with a .cbn file extension
    - Since this field data is read in from an external file instead of an output database, the ***From Step*** and ***From Frame*** boxes do not need to be filled in
4. Click ***OK*** to finish





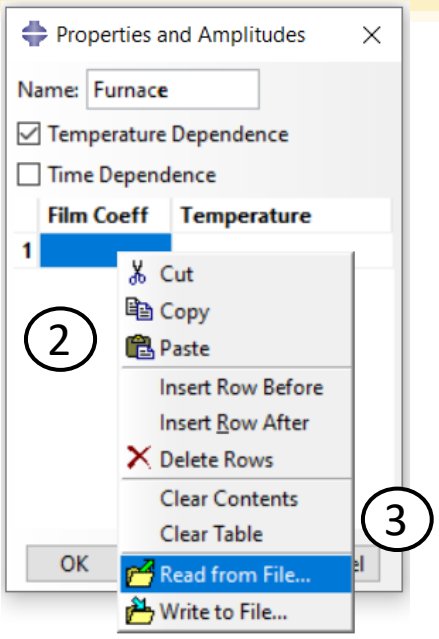


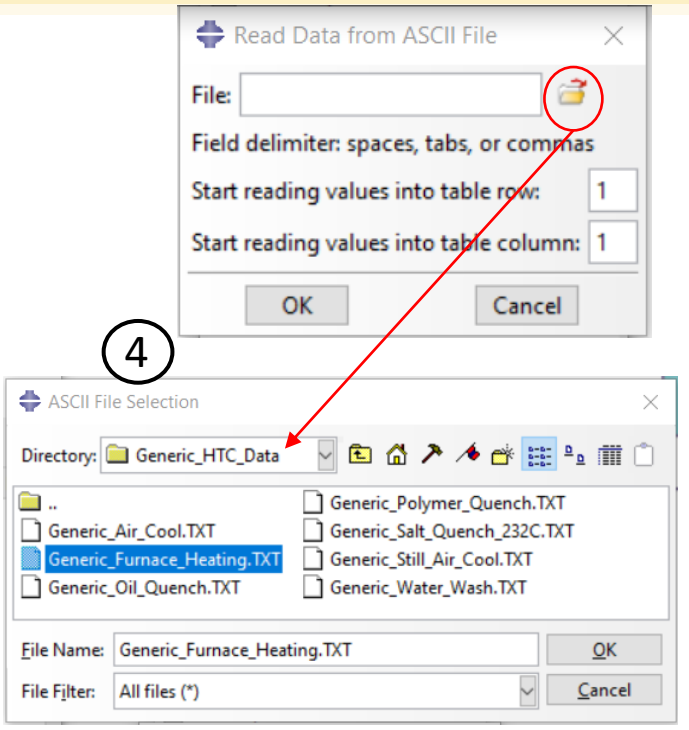


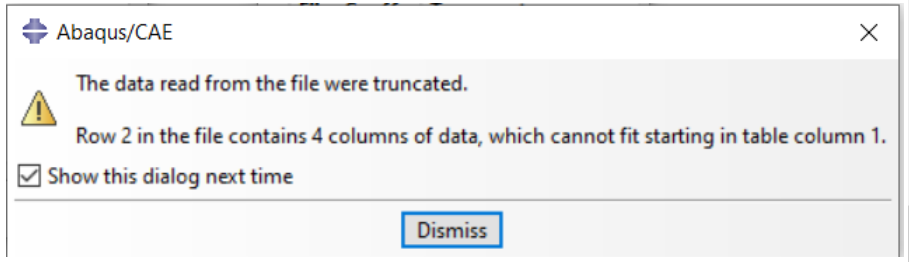
NOTE: You may get this error message if the number of characters in the file path to the external file exceeds 80 characters. In that case, you can simply rename the *.cbn file to a shorter name or move it to a directory closer to root

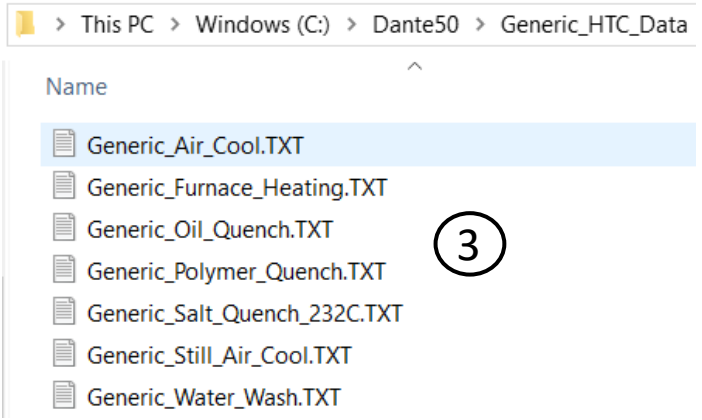
Step 5: Defining Film Properties

1. Click ***Add Film Property/Amplitude*** in the ***Initial/Boundary Conditions*** tab
2. Properties for the furnace, oil quenchant, and air cooling are defined; Give a name and check ***Temperature Dependence***
3. Right click on the first row and column and click ***Read from File***, film coefficient vs temperature information can be given in ascii text file format and some are in the DANTE parent directory in the ***Generic_HTC_Data*** folder
   - These lines can also be pasted into table or filled out line-by-line
4. Click ***OK*** to insert the data lines, a warning box may appear, click ***Dismiss***







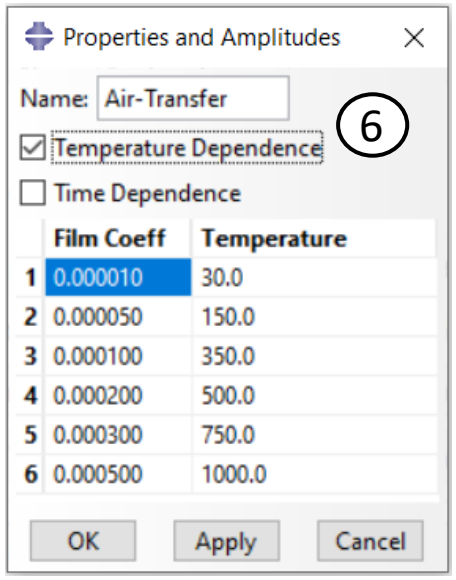


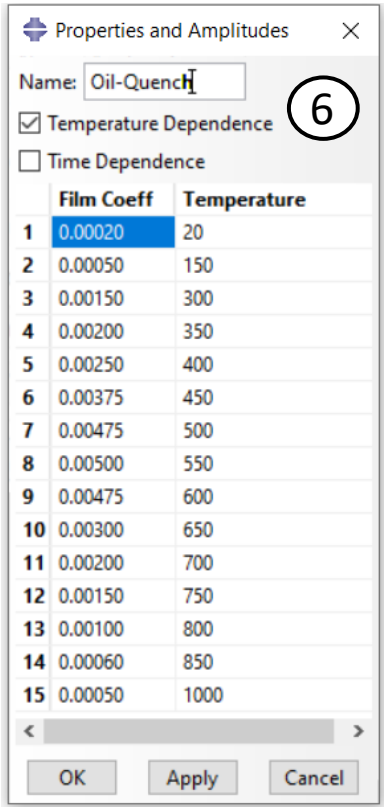
NOTE: DANTE comes with representative HTC data files which can be used or modified to fit specific cooling rates. Cooling rates differ from process to process, so experimentation may be needed to fit HTC values to a certain process.

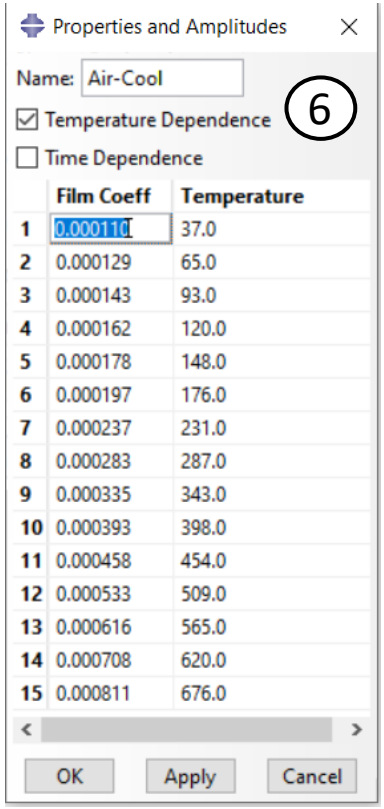
Step 5b: Defining Film Properties

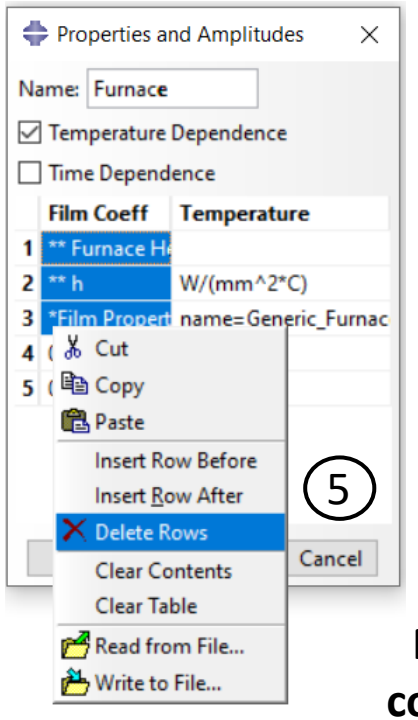
5. Some of the rows may contain extra information from the text file, highlight any rows containing non-values, right click and ***Delete Rows*** and click ***Apply***

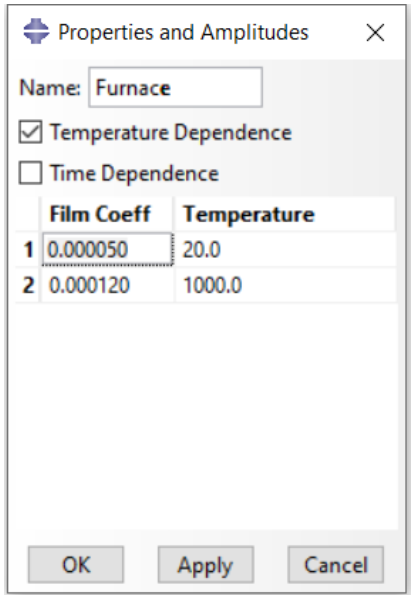
6. Data used from the **Generic_HTC_Data** folder for the following properties includes: **Generic_Furnace_Heating.TXT** for **Heat-Up**, **Generic_Still_Air.TXT** for **Air-Transfer**, **Generic_Oil_Quench.TXT** for **Oil-Quench**, and **Generic_Air_Cool.TXT** for **Air-Cool**

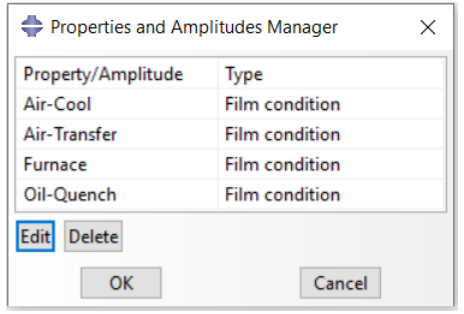








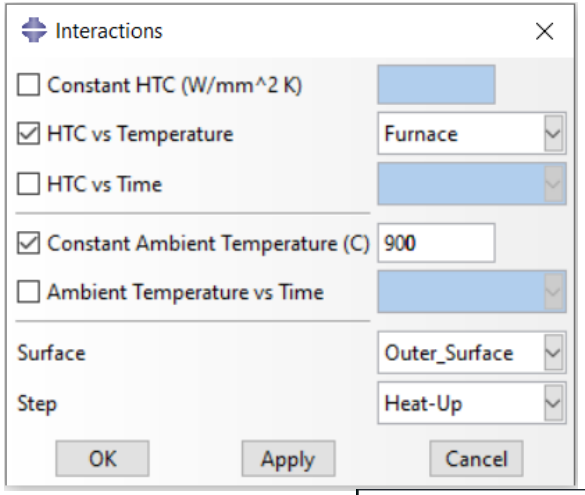


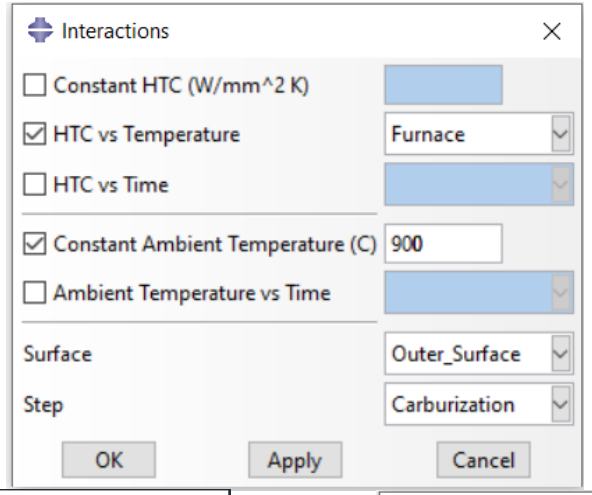


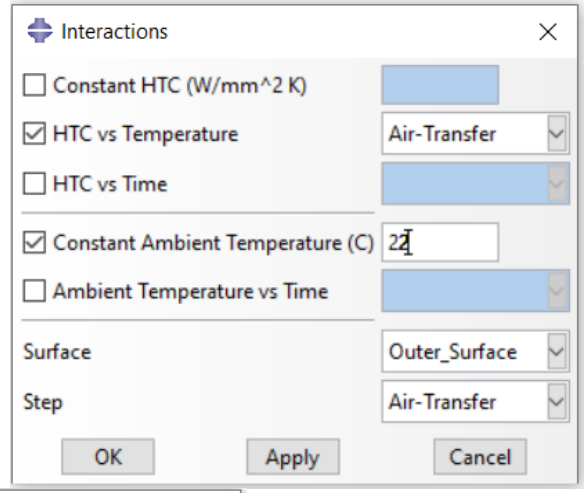
Note: These heat transfer coefficients can be specified as constant values by unchecking both dependences and giving a single value or directly in the Add Interaction dialog. They can be edited or deleted from the *Film Property/Amplitude Manager*

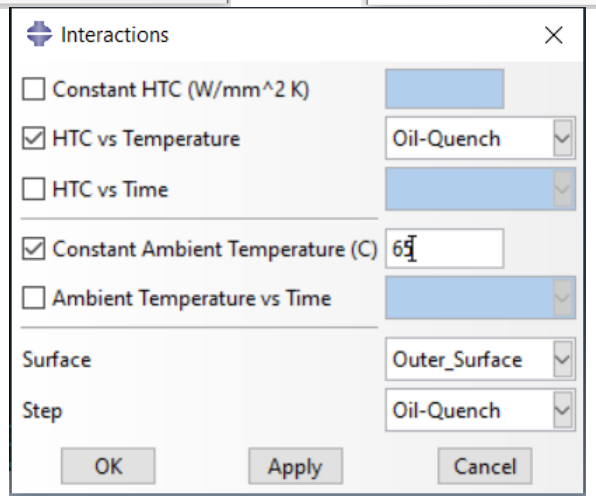
Step 6: Thermal Model Interactions

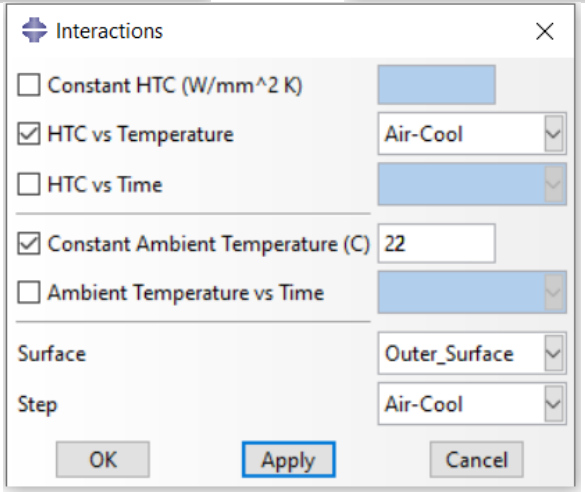
1. In the ***Initial/Boundary Conditions*** tab, click ***Add Interaction*** to now add the previously created film properties as interactions to the surfaces of the ring slice geometry at each respective step









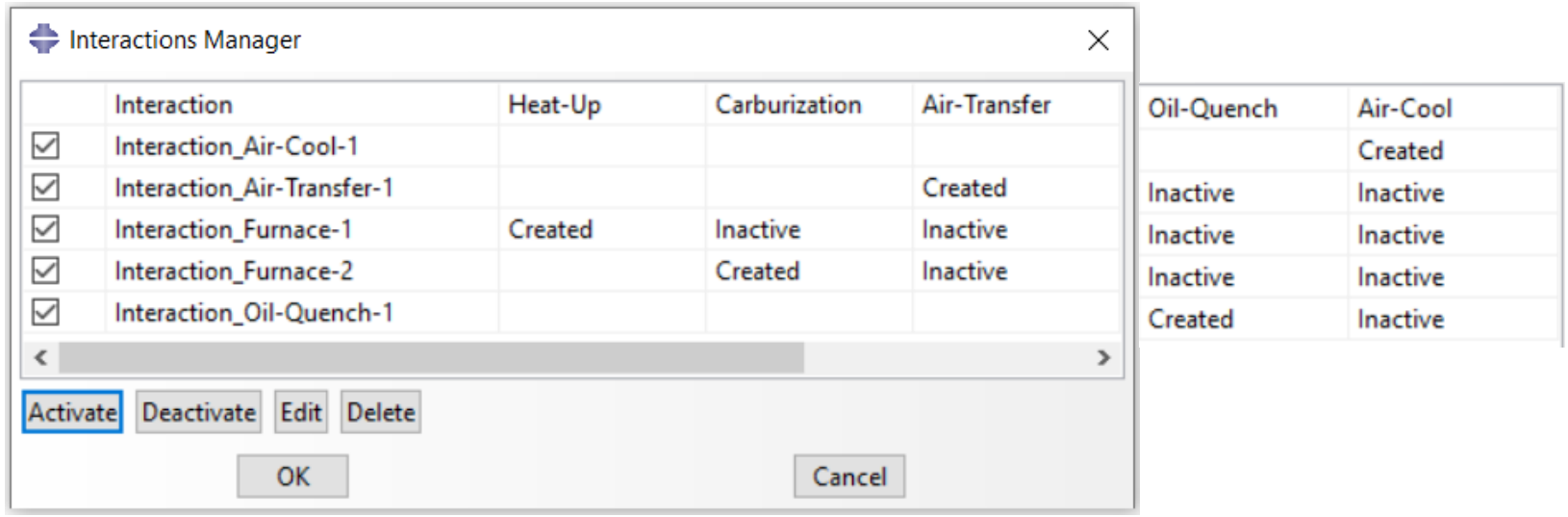


Each step must have an ambient temperature associated with it, here those values are constant

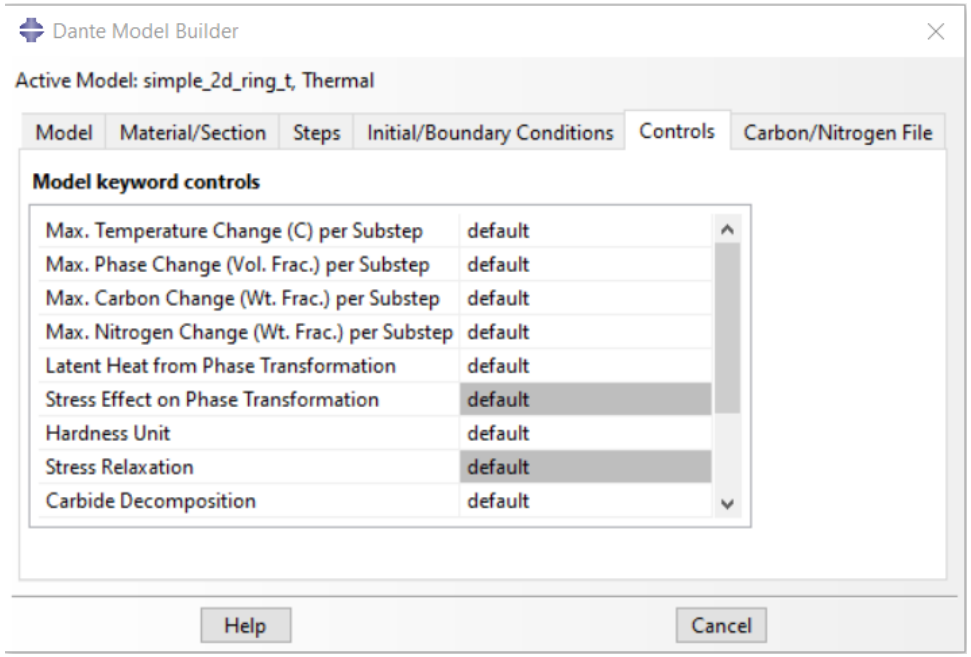
Step 6b: Thermal Model Interactions

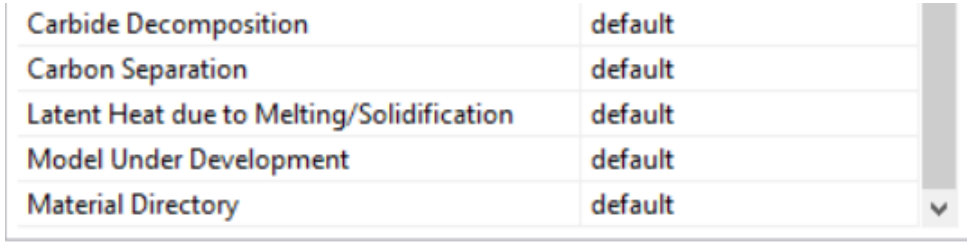
NOTE: Nothing needs to be changed on the table at this point, but it is a good idea to be aware of it.

- Please review the interactions to make sure they are inserted at the proper step ***AND*** that each step has an interaction associated with it, ***EVEN*** if that interaction is ***Propagated*** from another step
- By default, this Plug-In ***deactivates*** the interaction properties at all the steps in which they are not defined. Thus it is important to review each step to be sure that an interaction is properly defined or propagated
- Failure to do so might result in an error during simulation run time



Step 7: Thermal Model Dante Controls

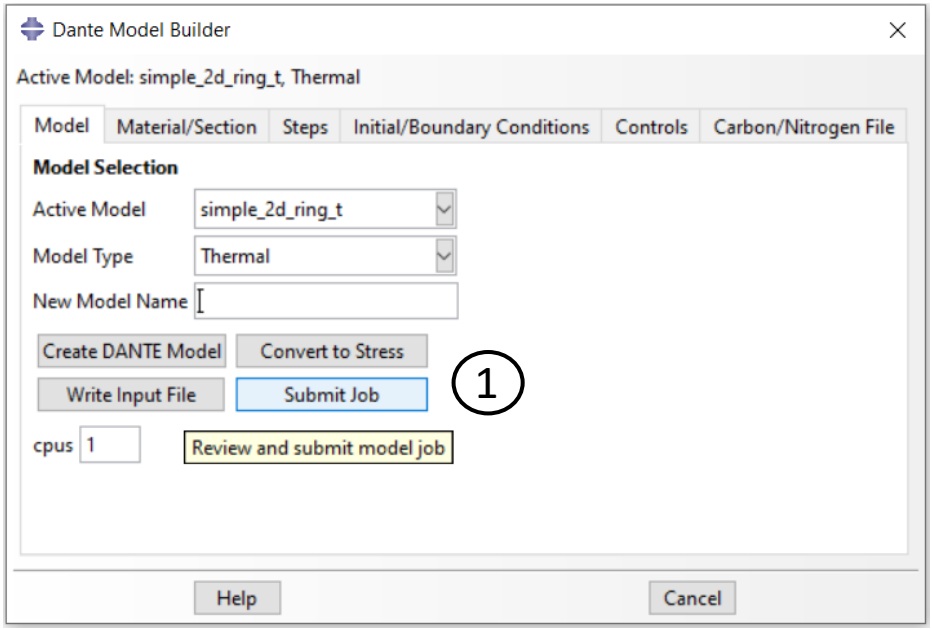




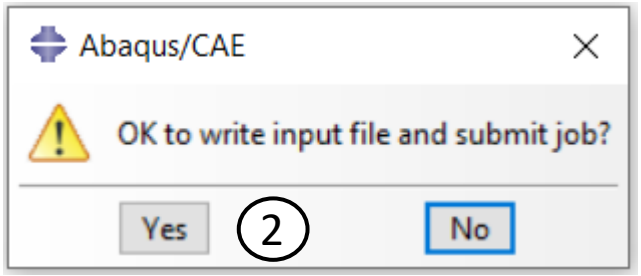
- The thermal model allows the user to change every DANTE control keyword in the ***Controls*** tab except for ***Stress Effect on Phase Transformation*** and ***Stress Relaxation*** since these two are available only in the stress model
- Everything on this page can be left as default

Step 8: Creating and Submitting Thermal Job

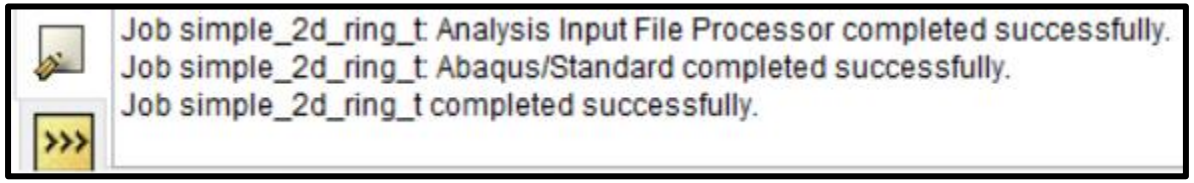
- Please refer to ***Step 16: Creating and Submitting Carburization Job*** for the steps on how to submit the job
- Once again, this thermal model input can be written out if desired or the job can be submitted directly



1. In the ***Model*** tab, click ***Submit Job***
2. Click ***Yes*** to write input file and submit job

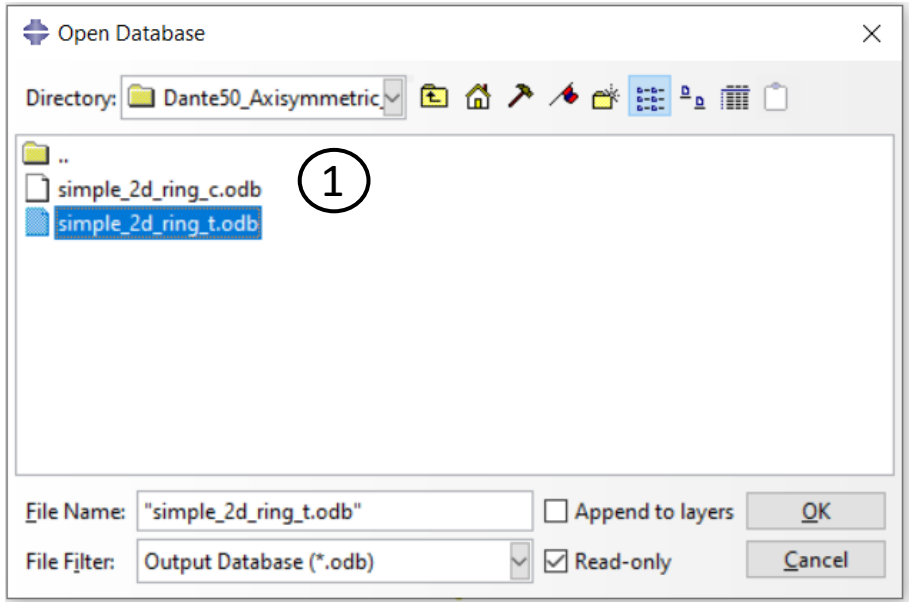


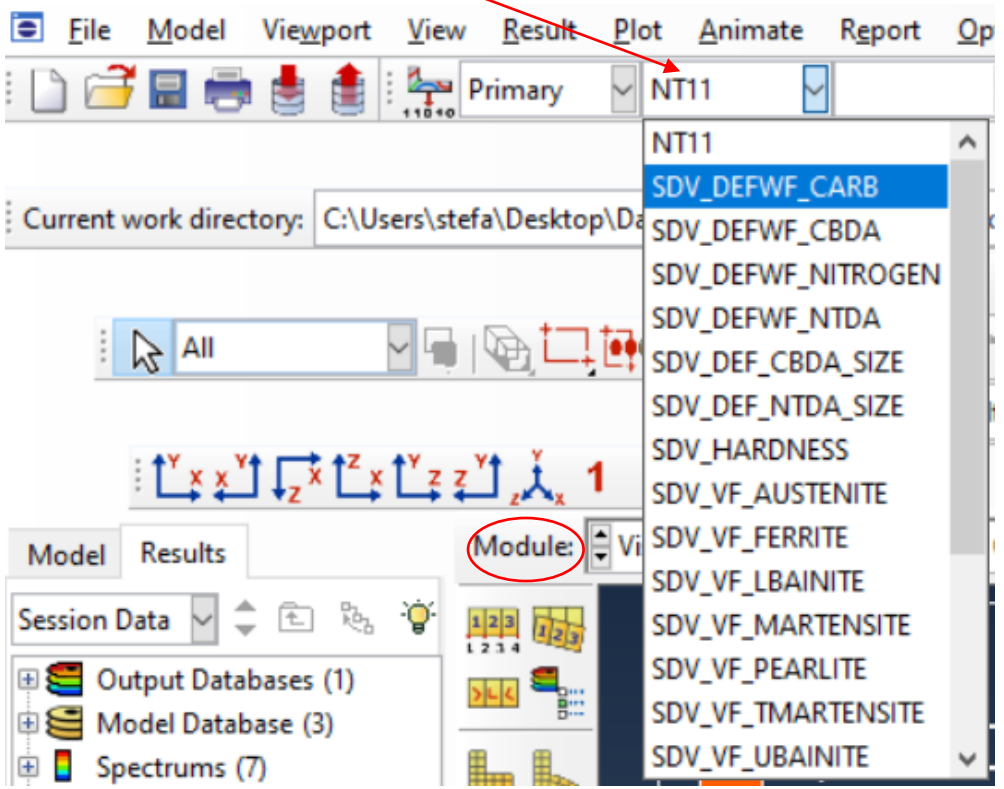
If the model run is successful, the lines should be displayed in the Abaqus CAE console

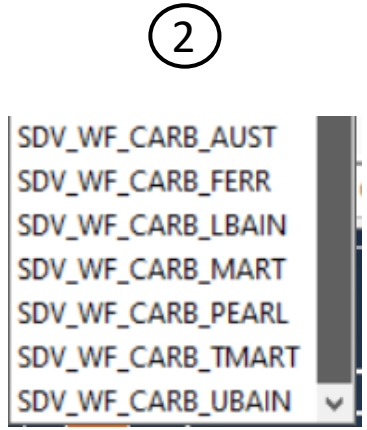


Step 9: Thermal Model Post-Processing

1. To open the thermal model output database, similar steps may be taken to ***Step 17: Carburization Model Post-Processing***
2. The **simple_2d_ring_t.odb** should be in the working directory and is the output database created from running the thermal model. It will open in the **Visualization** module and the dropdown menu in the CAE toolbar can be used to view results.



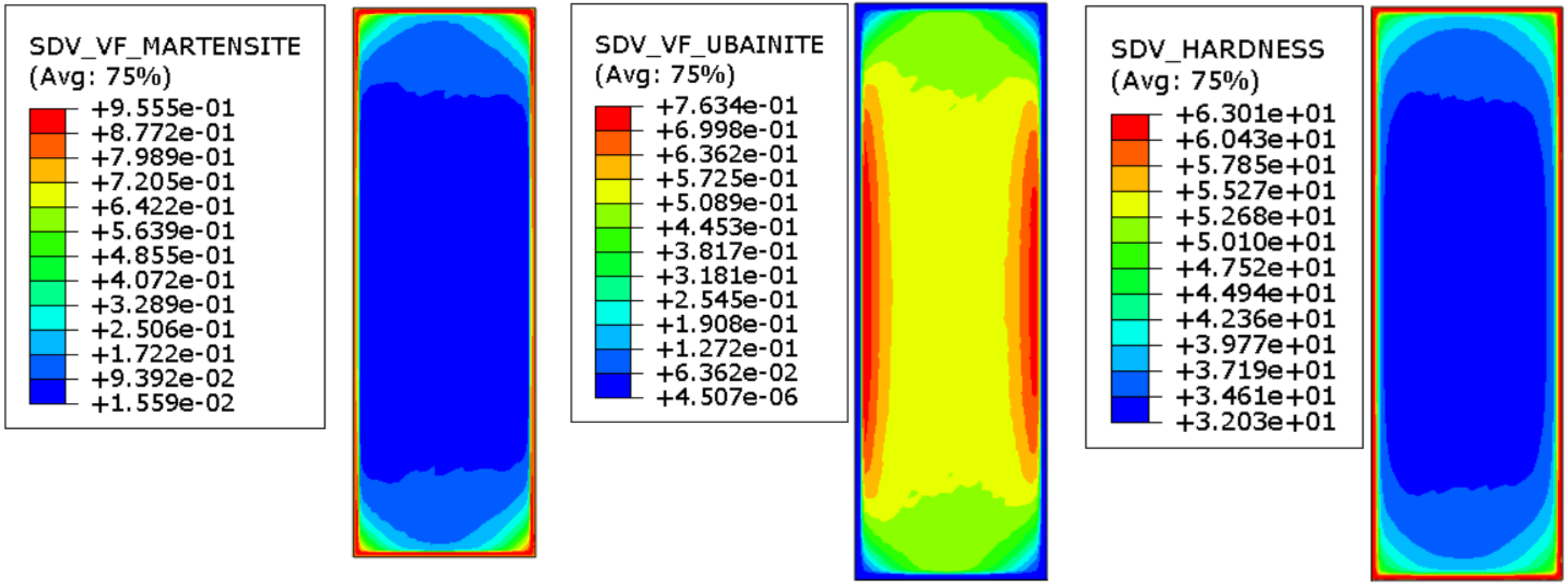




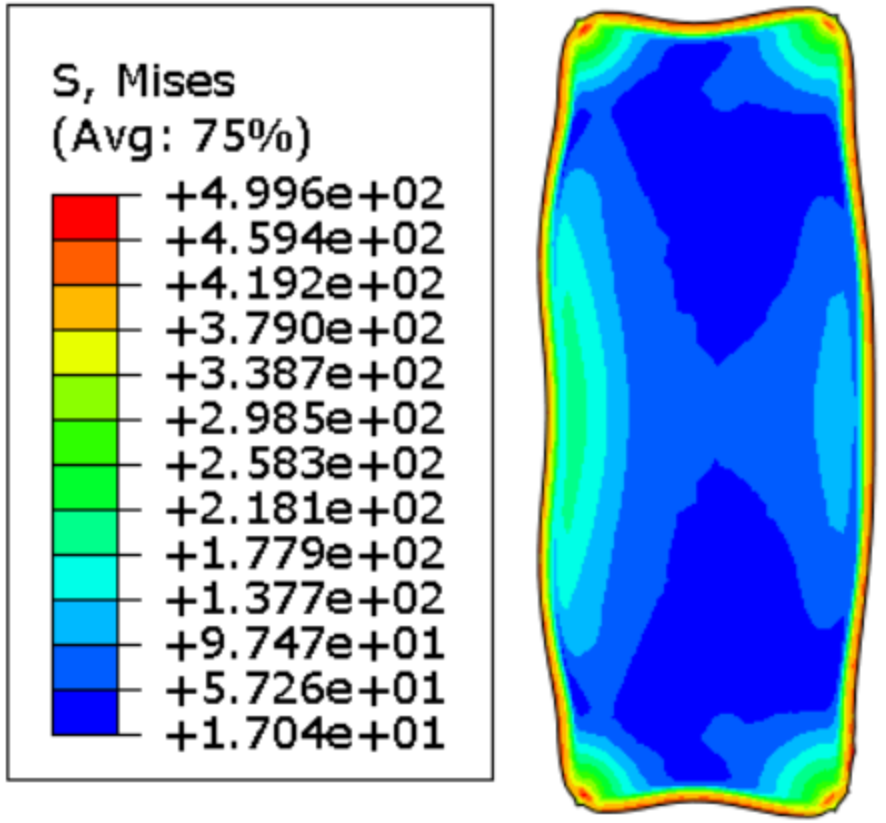
NOTE: Be sure to save the model at this point

Step 10: Visualizing Thermal Solution Dependent Variables

1. By selecting a solution variable, the contour image of the ring slice will be shown. By inspection, the ring slice has formed a martensitic case on the surfaces of the ring with high hardness and soft core in the middle

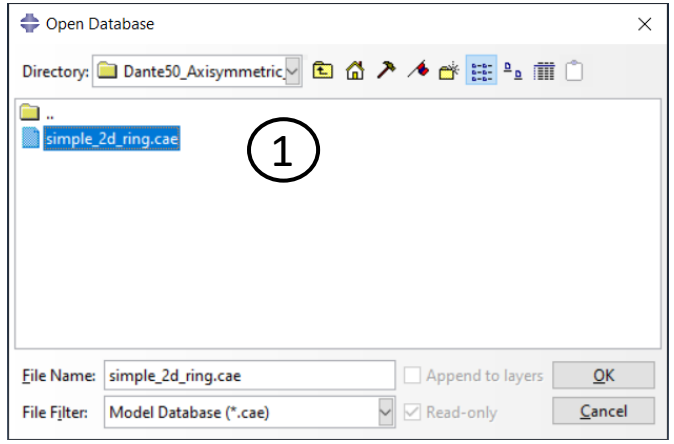


Stress Model

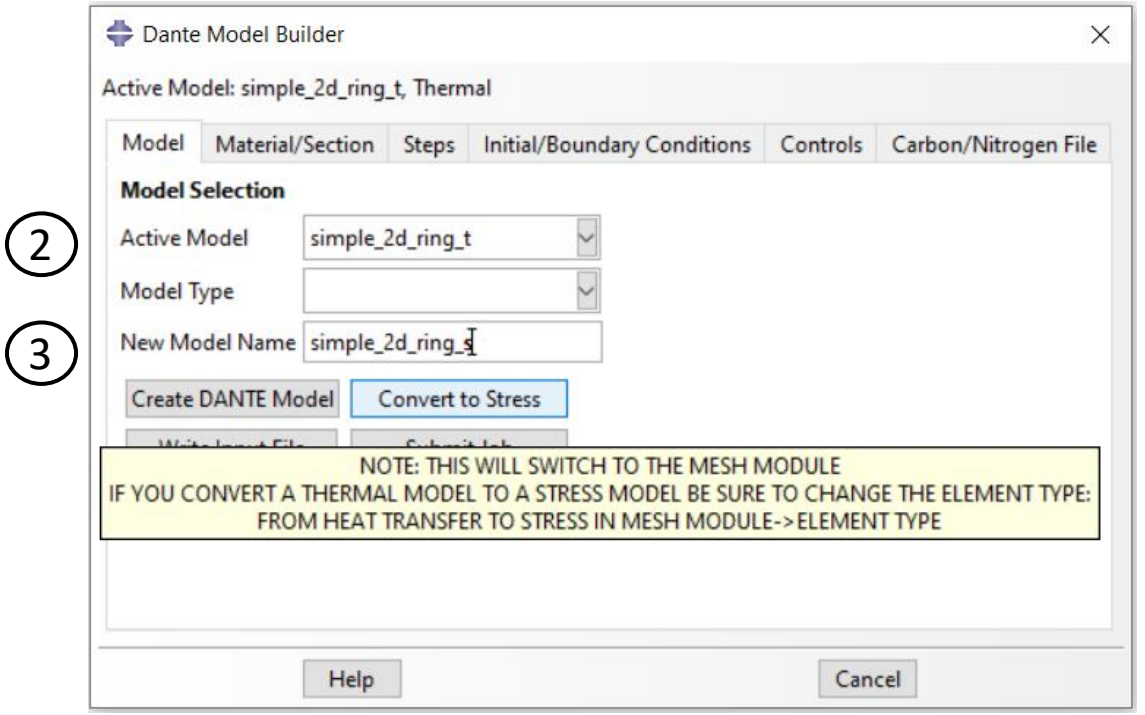


Step 1: Creating a Stress Model

- As with the previous Carbon and Thermal model, the Stress model may be built in a similar fashion
- However, the Dante Model Builder Plug-In provides the option to convert a thermal model to a stress model which is the method used here. This allows for quicker stress model set up

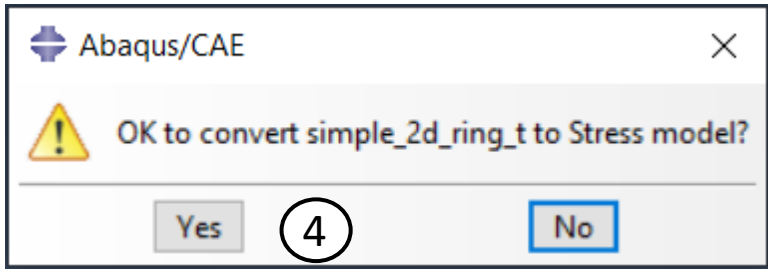


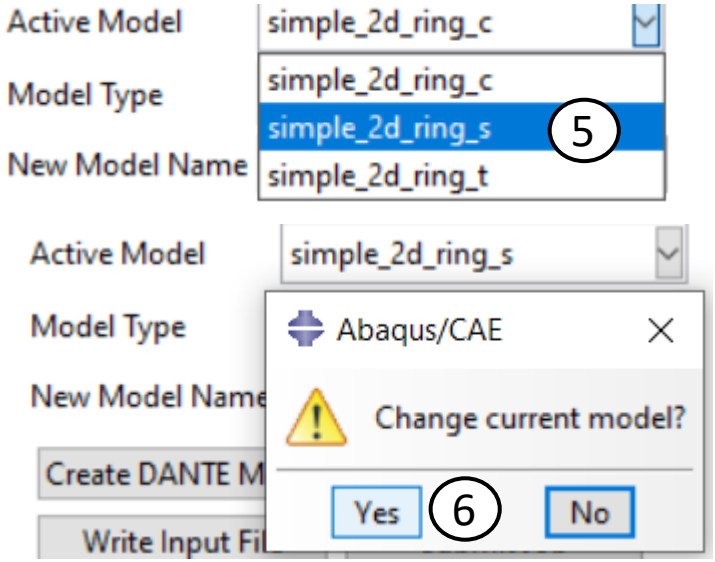
1. Open the Abaqus CAE and the CAE model used in this tutorial
2. Set the ***Active Model*** to the previous Thermal model
   - Note that when the Active Model is a Thermal model, the ***Convert to Stress*** button is no longer grayed out, meaning it can be used to convert an active thermal model into a basic stress model
3. Provide a ***New Model Name*** (simple_2d_ring_s)



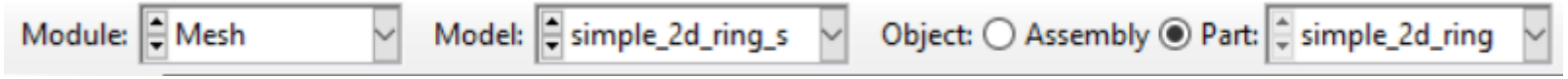
NOTE: If a name is not provided, the Active Model name will be used with an '_s' added to the end

Step 1b: Creating a Stress Model





4. Click the ***Convert to Stress*** button and ***Yes*** to confirm model conversion
   - If successful, this will create a DANTE stress model based on the thermal model by converting the heat transfer steps to static structural steps



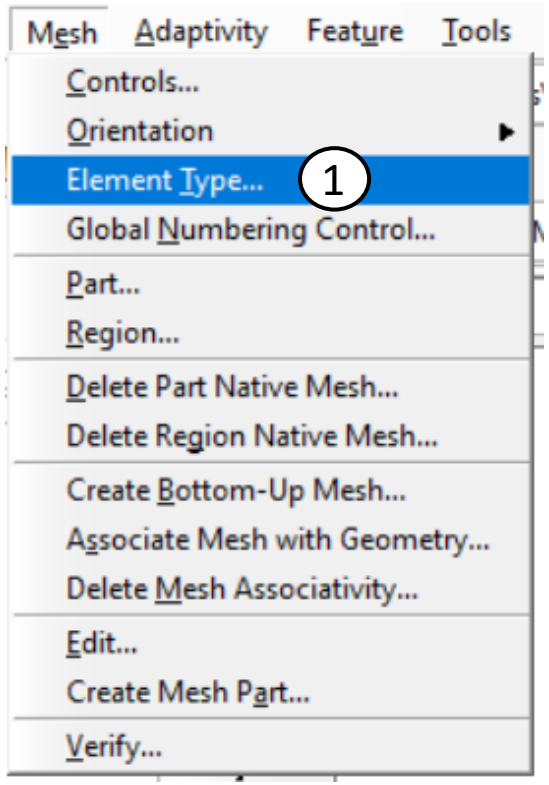
5. Since a new model is inserted into the model tree, the Plug-In must update, so set the ***Active Model*** to the newly created stress model
   - Note that the module should have changed to the ***Mesh*** module
6. Click ***Yes*** to switch to the stress model

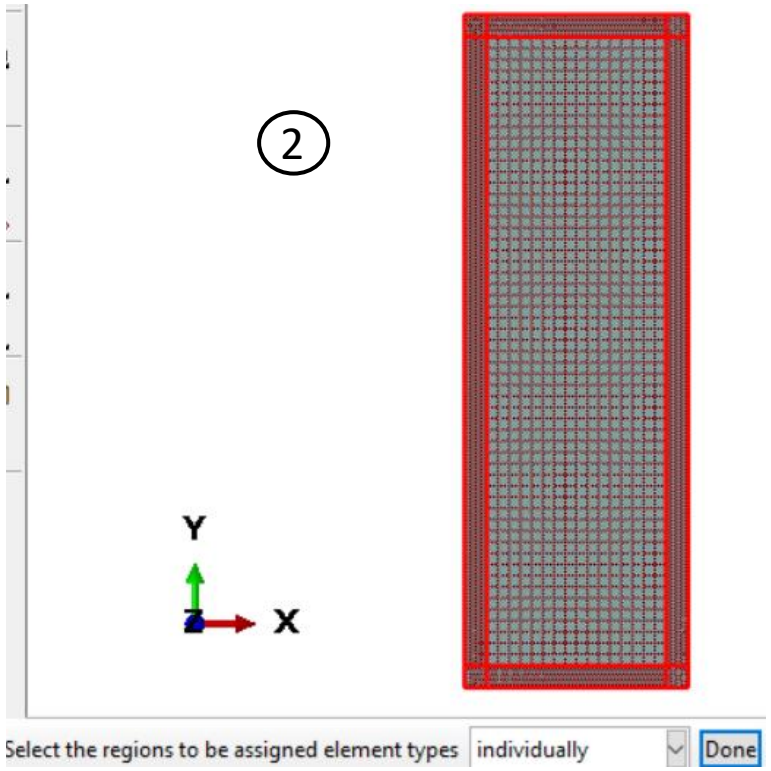
NOTE: The Convert to Stress feature is supposed to convert the relevant DANTE parameters for the stress model as well as recreate the thermal steps as stress steps. This saves the user time as they do not have to redefine the same steps in the stress model or insert the relevant DANTE stress model commands in the input file. What remains is to switch the mesh element type to stress elements and add basic stress displacement boundary conditions to the model

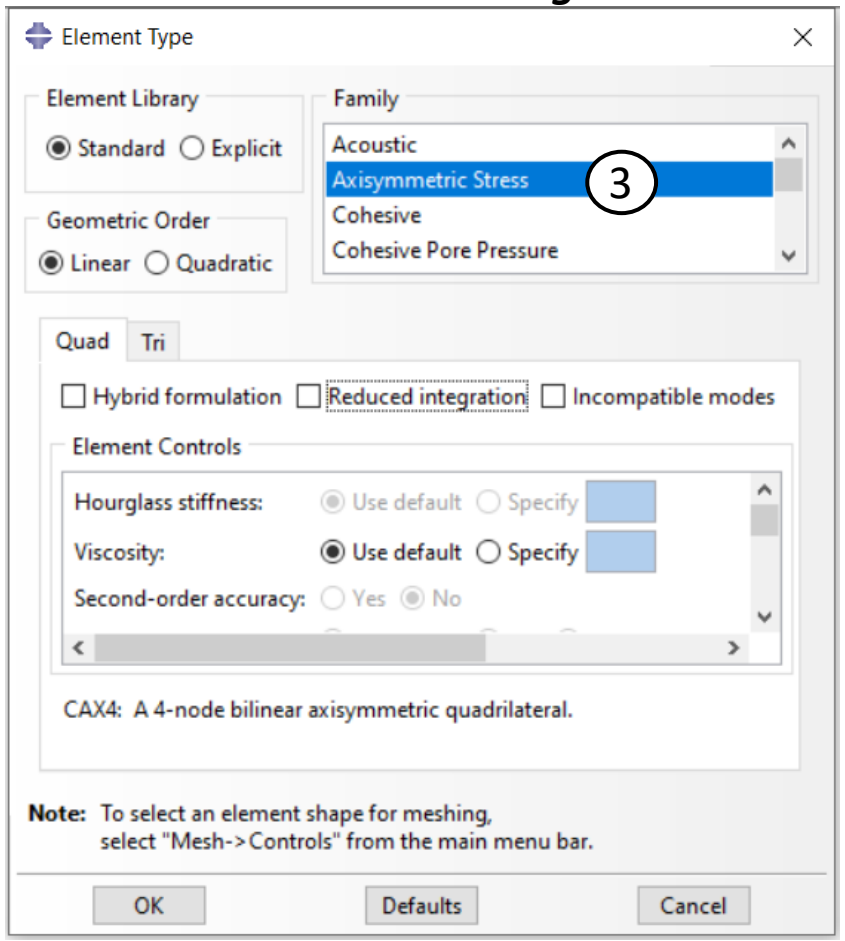
NOTE: Be sure to save the model at this point

Step 2: Converting Mesh to Stress Elements

1. The user must manually change the mesh type by selecting ***Mesh → Element Type***
2. Box select the entire ring slice and click ***Done***
3. In the Element Type dialog box, select ***Axisymmetric Stress*** in this case and uncheck the ***Reduced Integration*** box to use ***CAX4*** elements



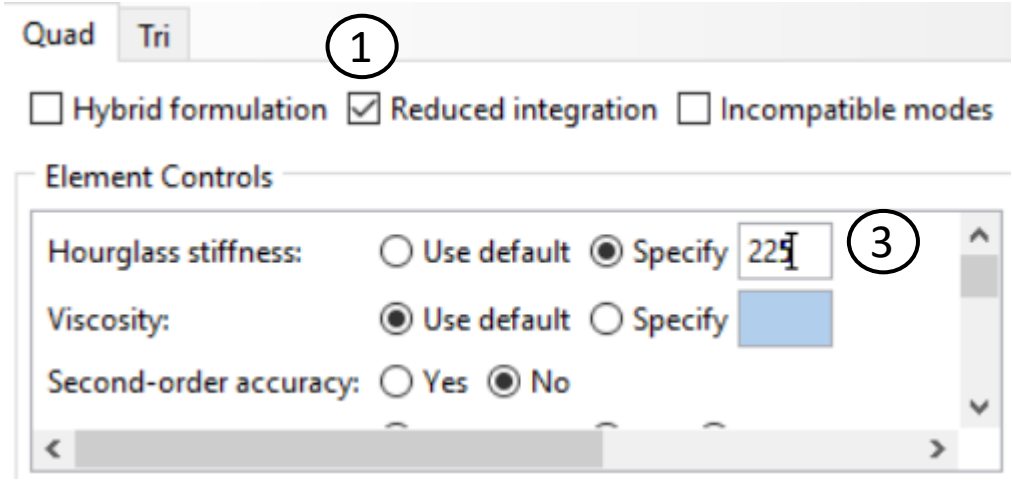


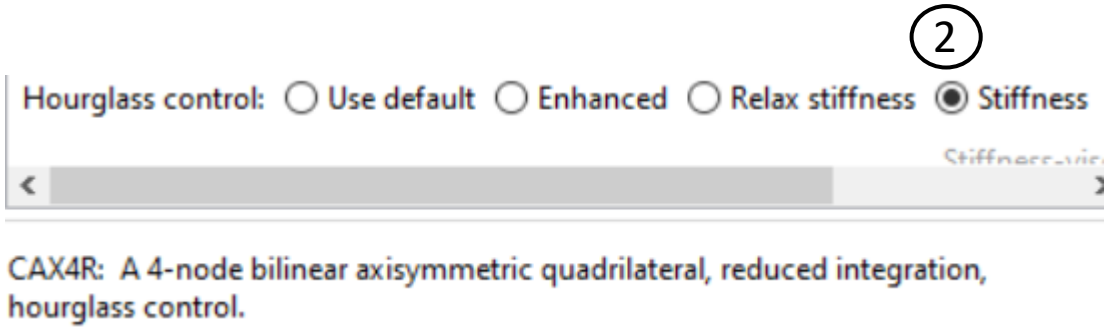


Step 3: Converting Mesh to Stress Elements

NOTE: If the CAX4 elements are used, this slide is irrelevant.

1. If you wish to use reduced integration points make sure to check the box
2. Then click on ***Stiffness*** in the ***Hourglass control*** section
3. At the top again, the ***Hourglass stiffness*** can be specified, so set it to ***225***, this value should work for most cases

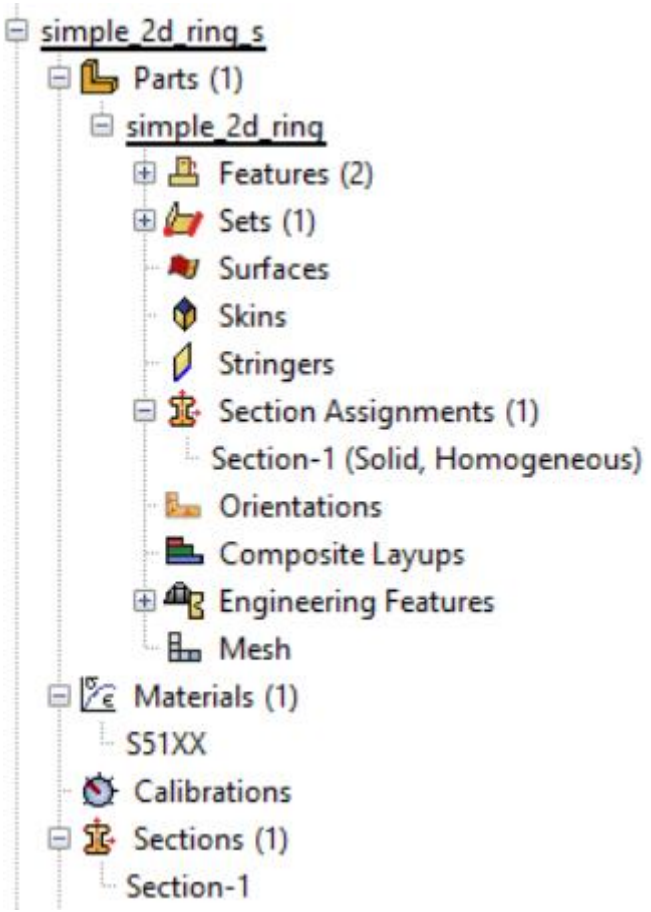


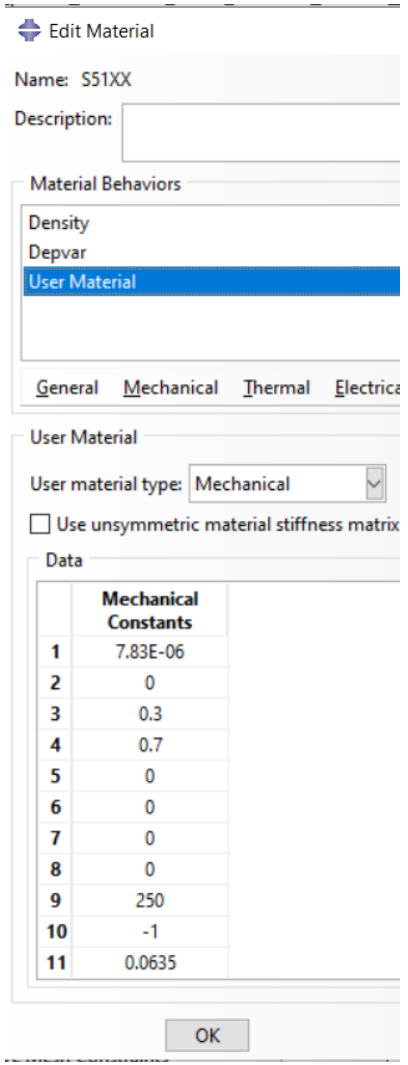


Step 4: Reviewing the Material

- The material, section, and section assignment should have been preserved from the thermal model and the material properties should have been converted to Mechanical material properties

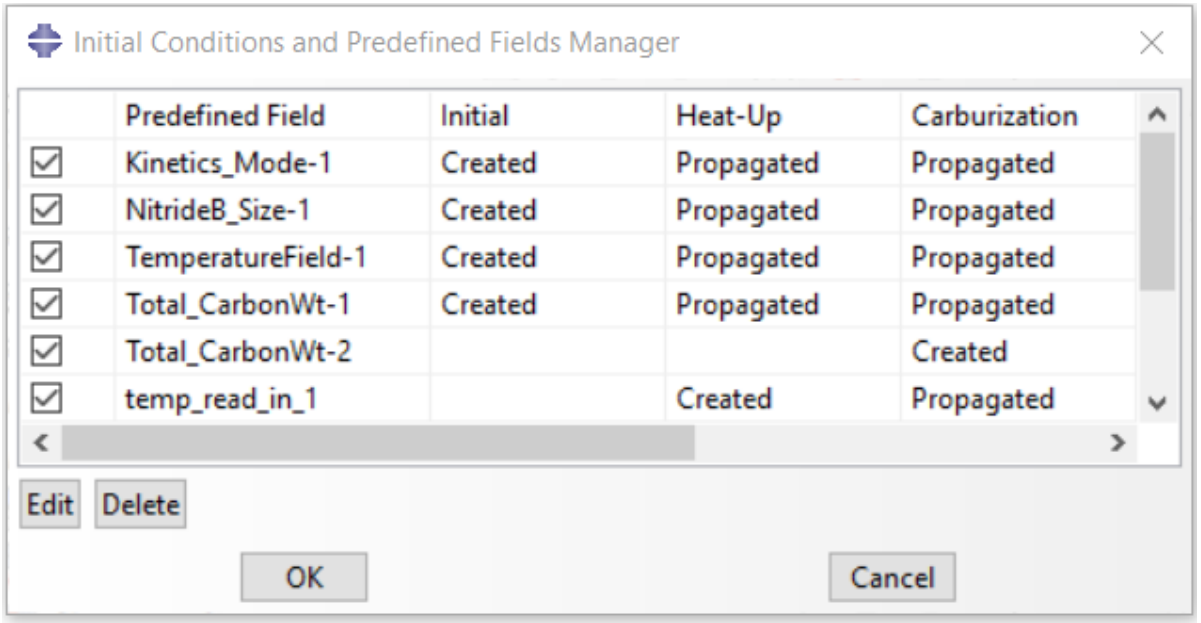
- Double click the material S51XX under the simple_2d_ring_s model tree to view the material definitions if desired. Clicking User Material will show that the material constants are now defined under the Mechanical material type
- When defining material for a stress model from scratch, the plug-in Material/Section tab may be used to insert the DANTE material information into the CAE model directly or it can be defined manually under the model tree

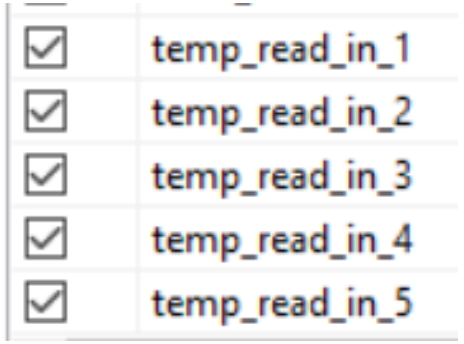




Step 5: Reviewing Initial Conditions and Predefined Fields

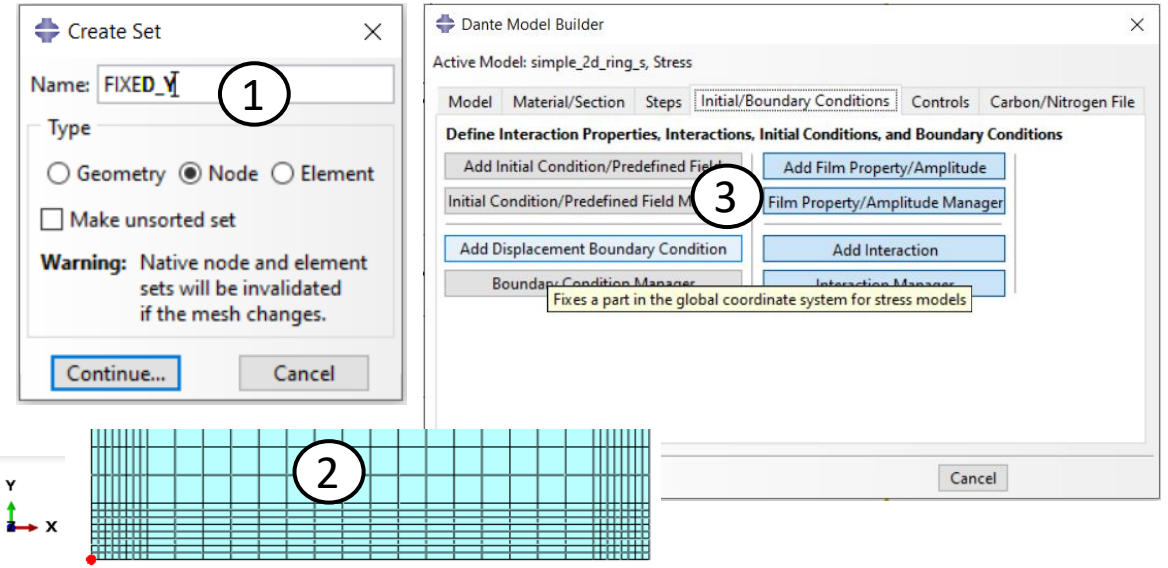
1. Back in the Plug-In, review the ***Initial Condition/Predefined Field Manager*** table, the temperature profile should have been mapped into the stress model from the thermal output database at each step
   - If they are not there or if a stress model is being built independently of the thermal model, create these temperature predefined fields from the ***Add Initial Condition/Predefined Field*** box to specify the External Data from an external file or output database and setting the step and step increment to retrieve the temperature information
   - All other predefined fields should be available as well

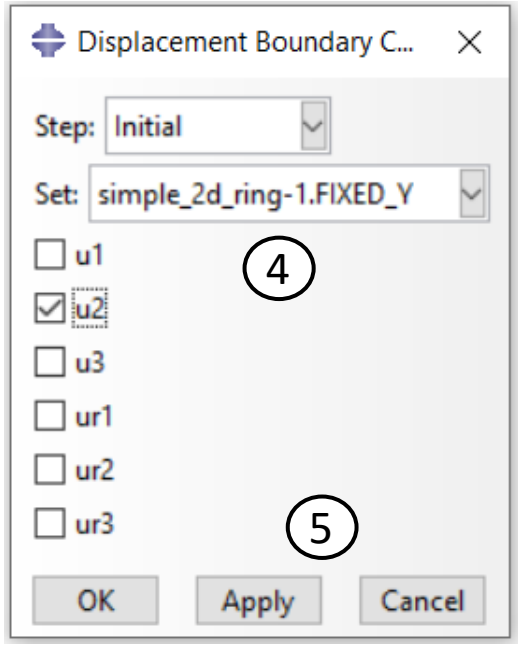


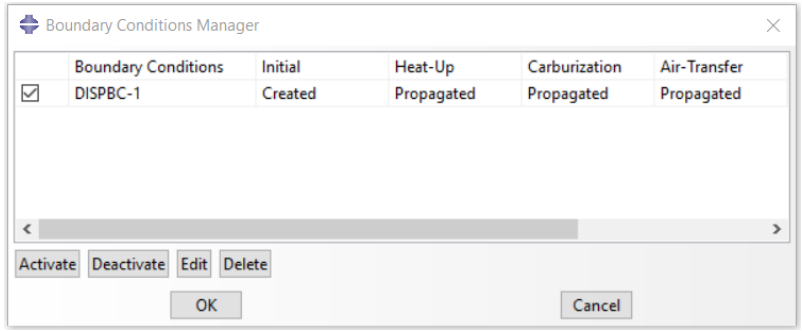


Step 6: Defining the Displacement Boundary Conditions

1. To fix this axisymmetric model, a single node must be fixed so a node set must be defined in the **Assembly** module → ***Tools*** → ***Set*** → ***Create*** and give a name and select ***Node*** and ***Continue***
2. Select the node on the corner at the inner diameter of the ring slice and click ***Done***
3. Go to ***Initial/Boundary Conditions*** tab → ***Add Displacement Boundary Condition***
4. Let the Step be 'Initial', choose the node set defined above, and check ***u2*** so that the node does not move in Y-dir, thus fixing the 2D axisymmetric part while allowing it to grow and shrink due to the thermal and phase transformation effects
5. Click ***OK*** to finish the boundary condition, this can be reviewed and edited in the ***Boundary Conditions Manager***







NOTE: Since this is an axisymmetric model, ONLY u1, u2, and ur3 can be selected

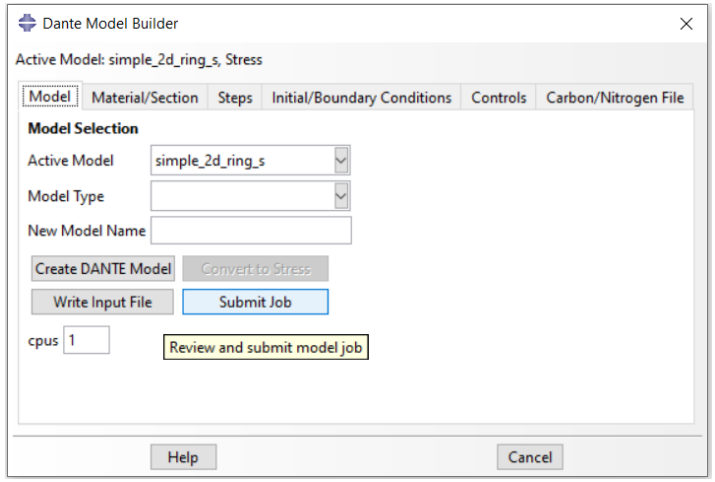
Step 7: Reviewing the Stress Model Dante Controls and Submitting the Job

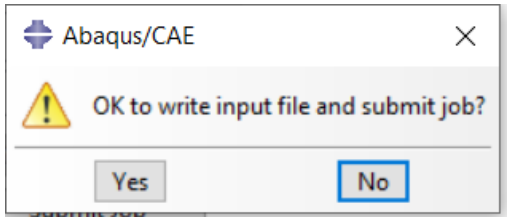
- The DANTE controls can also be reviewed, no changes will be made here, but notice how the ***Latent Heat from Phase Transformation*** and ***Latent Heat due to Melting/Solidification*** are read-only

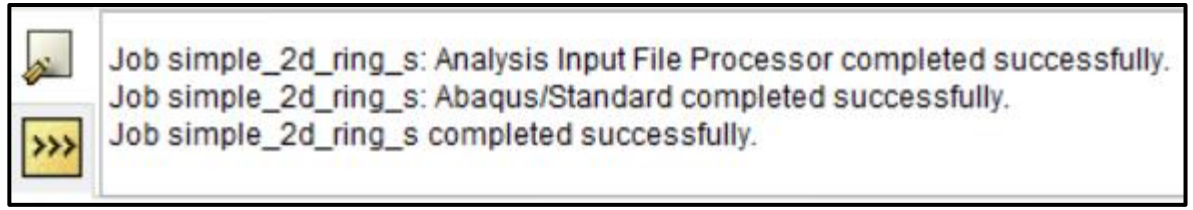
- This concludes the stress model setup, be sure to save the model, go to the ***Model*** tab and click ***Submit Job*** or ***Write Input File*** as was done previously

Model keyword controls

Latent Heat from Phase Transformation	default
Stress Effect on Phase Transformation	default
Hardness Unit	default
Stress Relaxation	default
Carbide Decomposition	default
Carbon Separation	default
Latent Heat due to Melting/Solidification	default
Model Under Development	default
Material Directory	default







NOTE: If the model runs successfully, the following messages should appear in the CAE console

Step 8: Stress Model Post-Processing

1. Open the stress model output database with ***File → Open***
2. From the dropdown, select a stress state to view such as Mises
3. Displacement as well as all the other DANTE solution dependent variables can be viewed as well
4. Under ***Options → Common***, change the deformation scale factor to 1 to view actual deformed shape and select ***Feature edges***