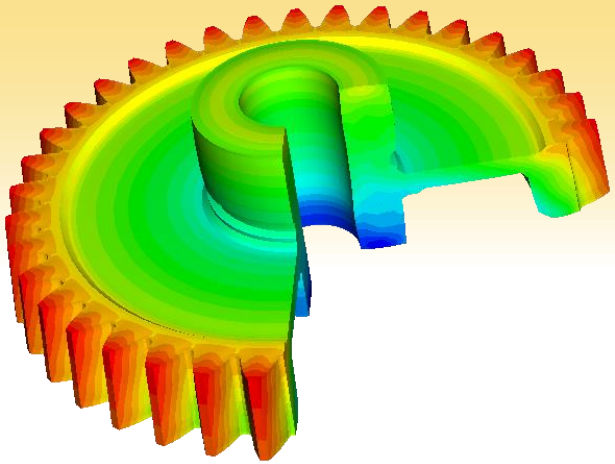
Dante 5.0 Tutorial

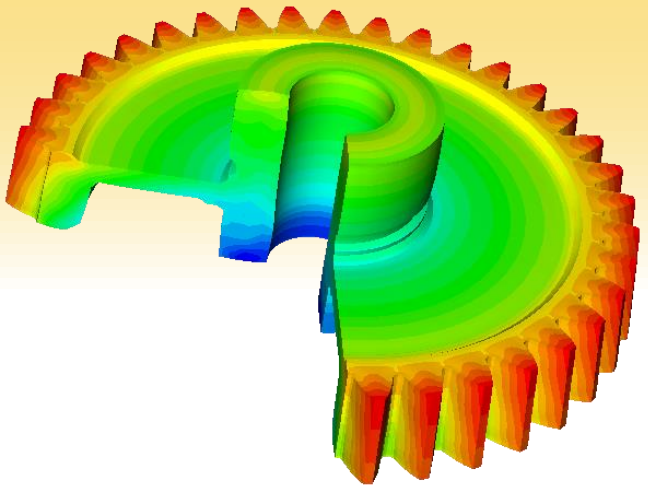
Axisymmetric Simple 2D Ring Model

Prepared By

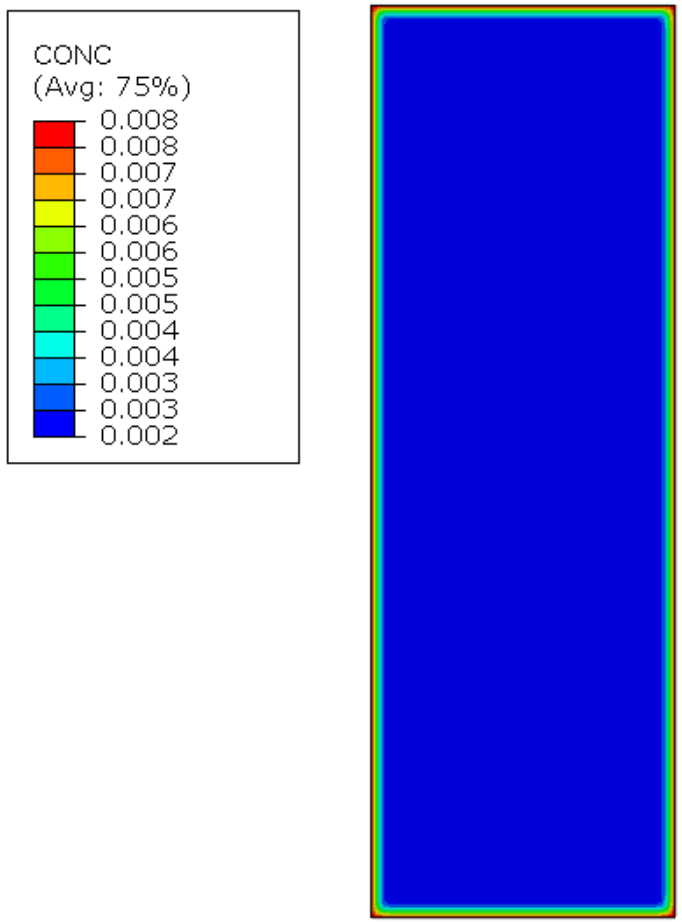
DANTE Solutions, Inc.

Cleveland Oh



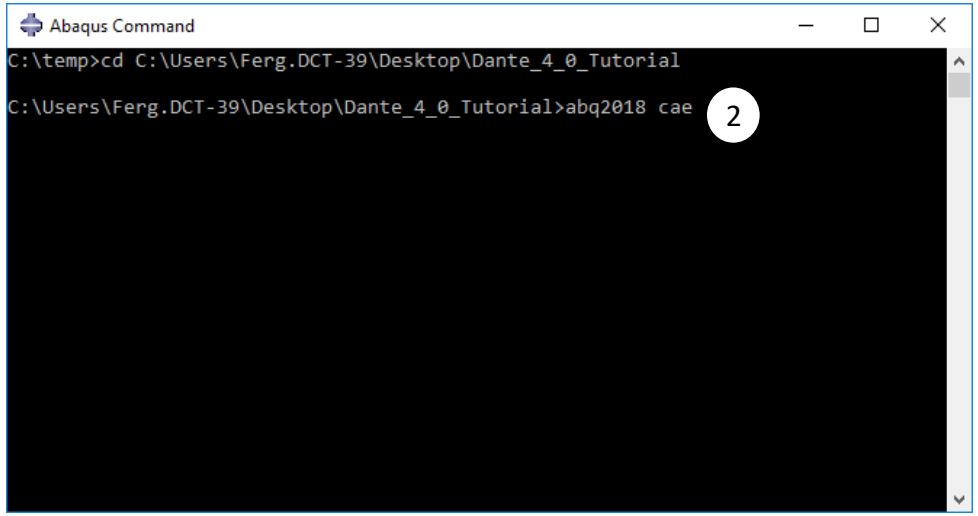


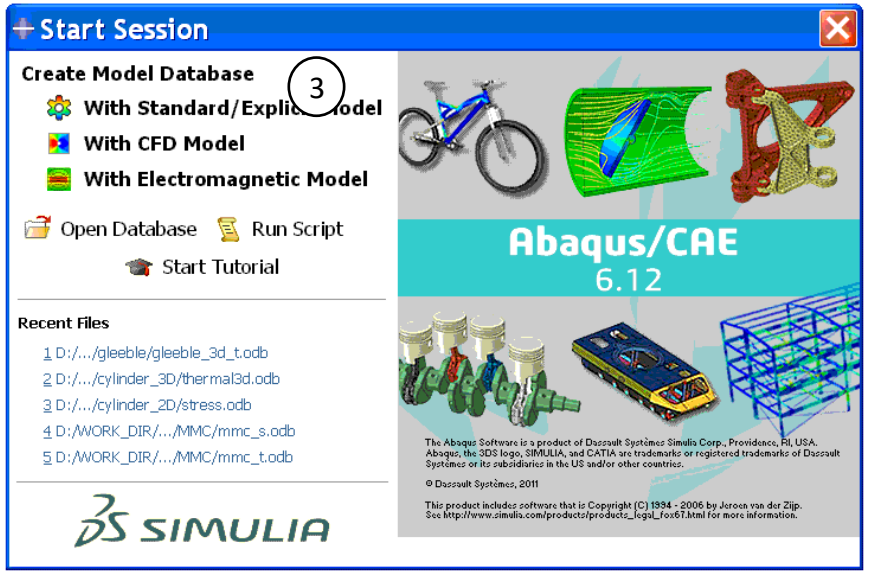
Carburization Model



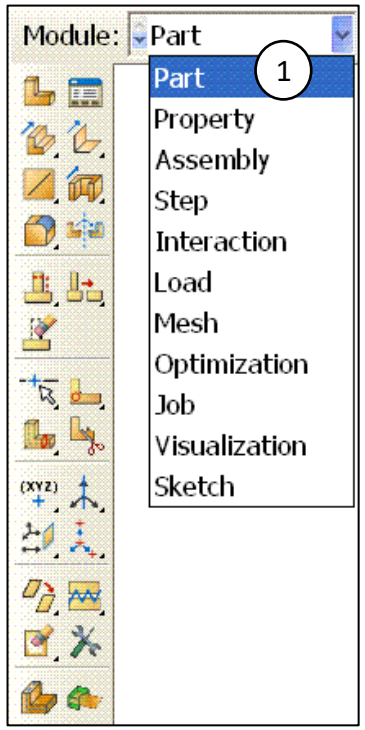
Step 1: Start Abaqus/CAE

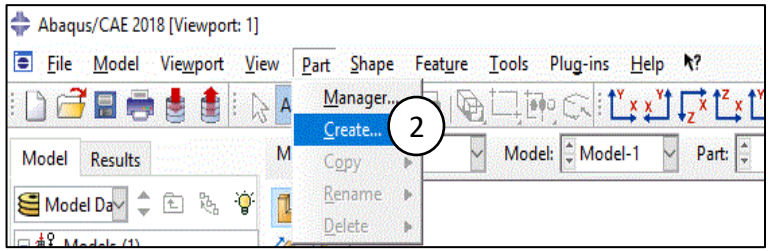
1. Open Abaqus Command and set the working directory by using the command ***"cd NameOfDirectoryLocation"***
2. Start Abaqus CAE by typing in the command "***abqxxxx cae***" into the command window
   - xxxx represents the version of Abaqus to be opened
3. Click ***With Standard/Explicit Model*** to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



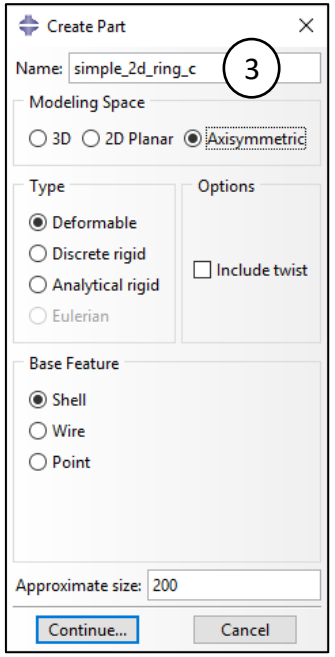


Step 2: Creating a Part

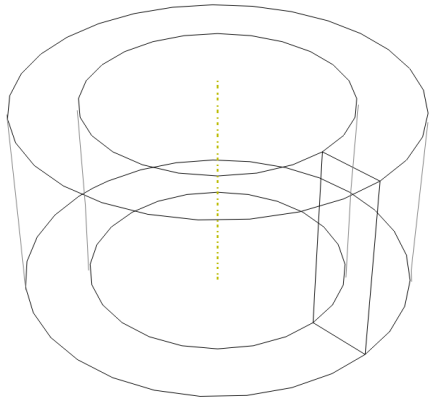


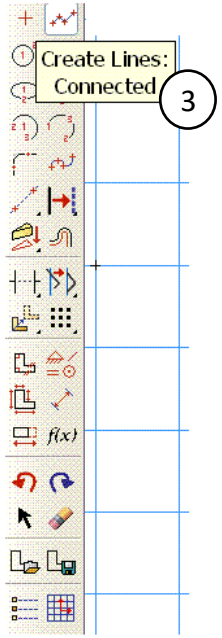


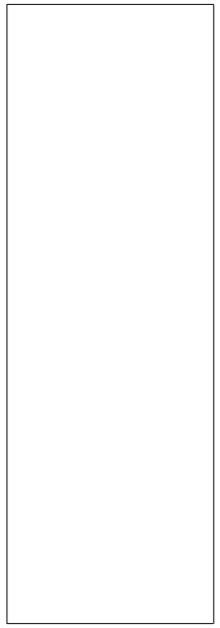
1. Under ***Module*** select ***Part*** to create a geometry
2. In the pulldown menu, select ***Part → Create***
3. Rename the part ***simple_2d_ring_c*** and select ***Axisymmetric***, ***Deformable***, ***Shell***, and set the approximate size to ***200*** then click continue



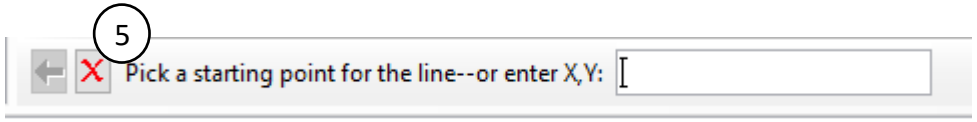
Step 2b: Creating a Part



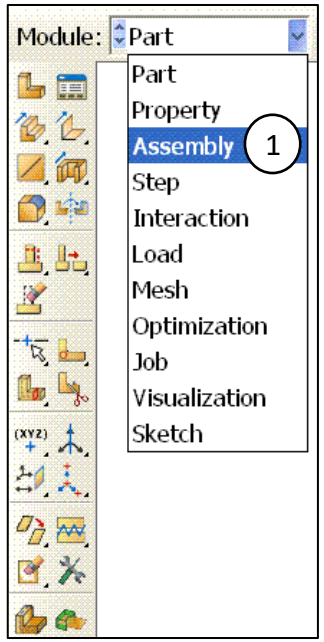


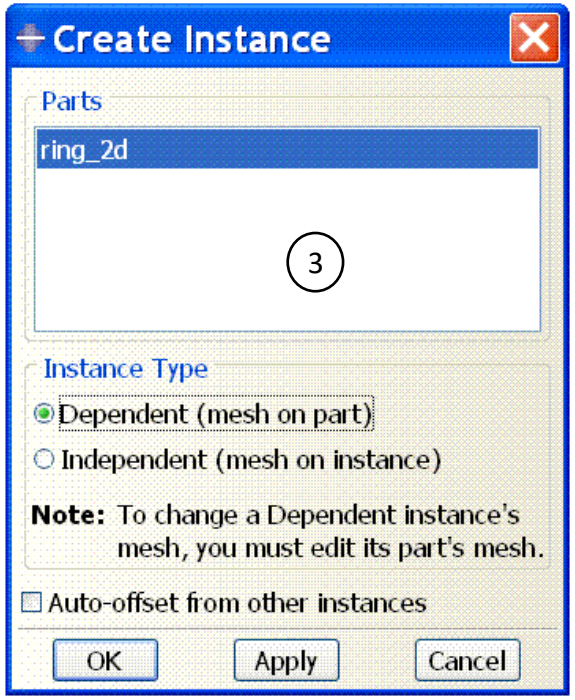


1. The ring has an inner diameter of 80 mm, outer diameter of 120 mm, and a height of 60 mm
2. An axisymmetric rectangular cross section will be used to model the ring
3. Select the ***Created Lines: Connected*** tool and type in (40,0), (60,0), (60,60), and (40,60)
4. Connect the last side by using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



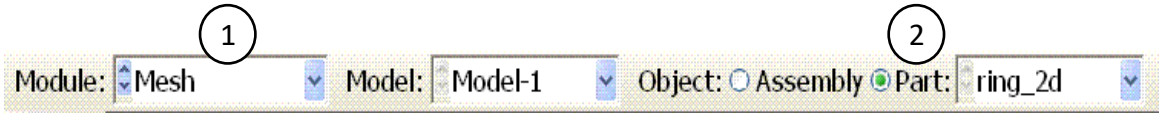
Step 3: Create an Instance

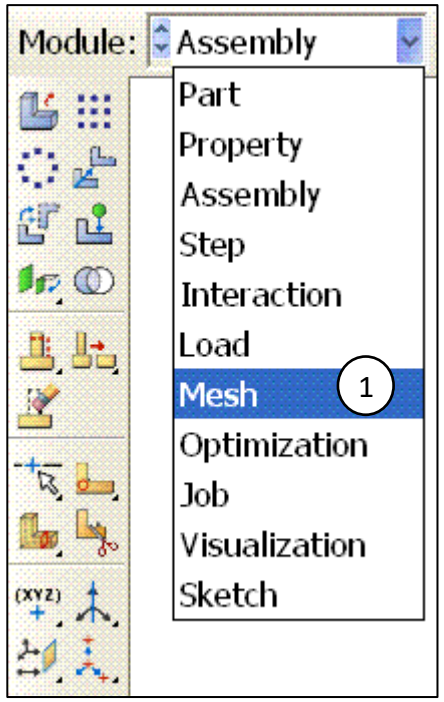


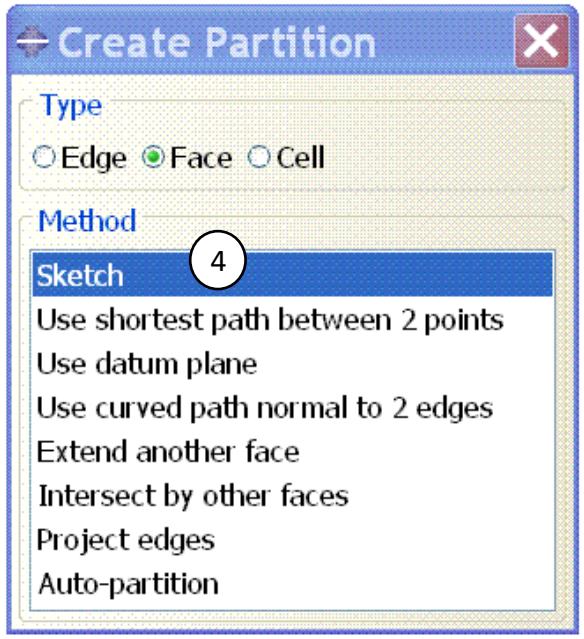


1. Under ***Module***, select ***Assembly*** to create the geometry
2. In the pulldown menu, select ***Instance → Create***
3. Select the part to be instanced and check ***Dependent (mesh on part)***
4. Click ***OK*** and the part should turn blue

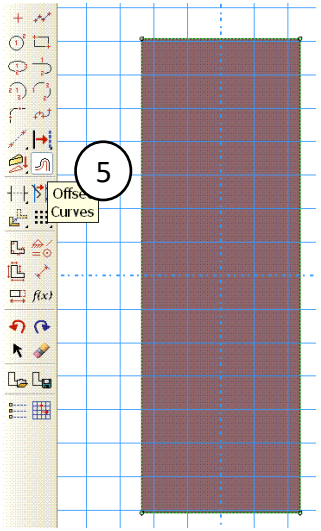
Step 4: Meshing the Part

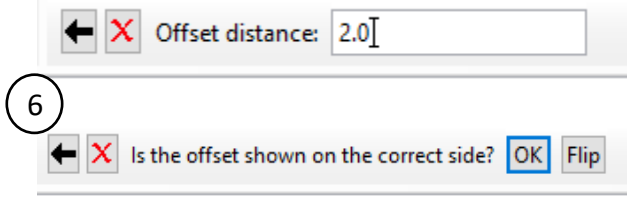


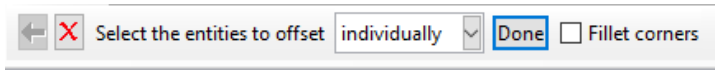




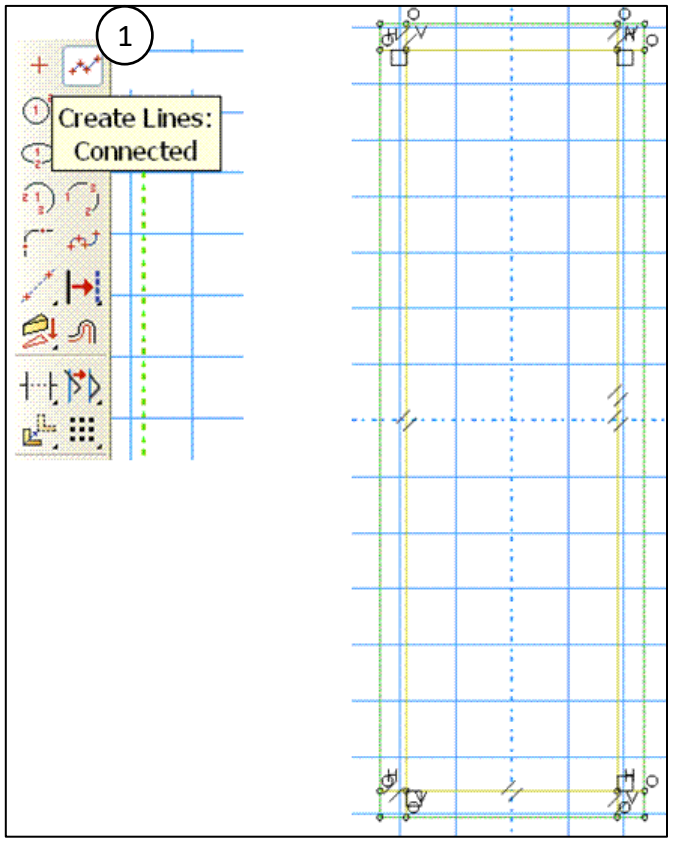
1. Under ***Module,*** select ***Mesh*** to mesh the part
2. Check ***Part*** instead of ***Assembly***
3. In the pulldown menu, select ***Tools → Partition***
4. Check ***Face*** under Type and select ***Sketch*** *under Method*
5. Select ***Offset Curves ,*** select all the sides, then click **Done**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

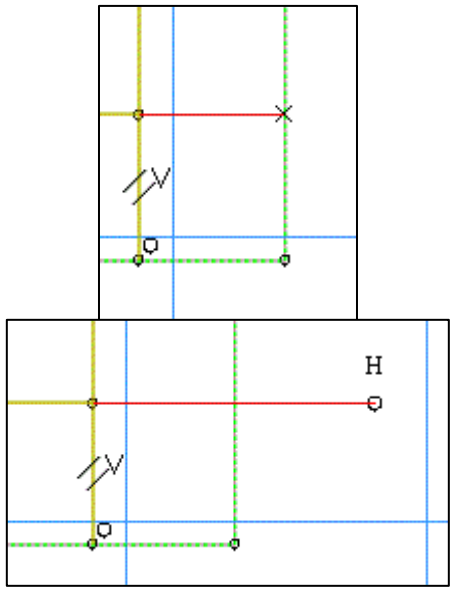




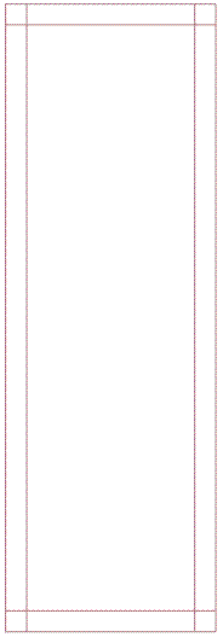


Step 4b: Meshing the Part

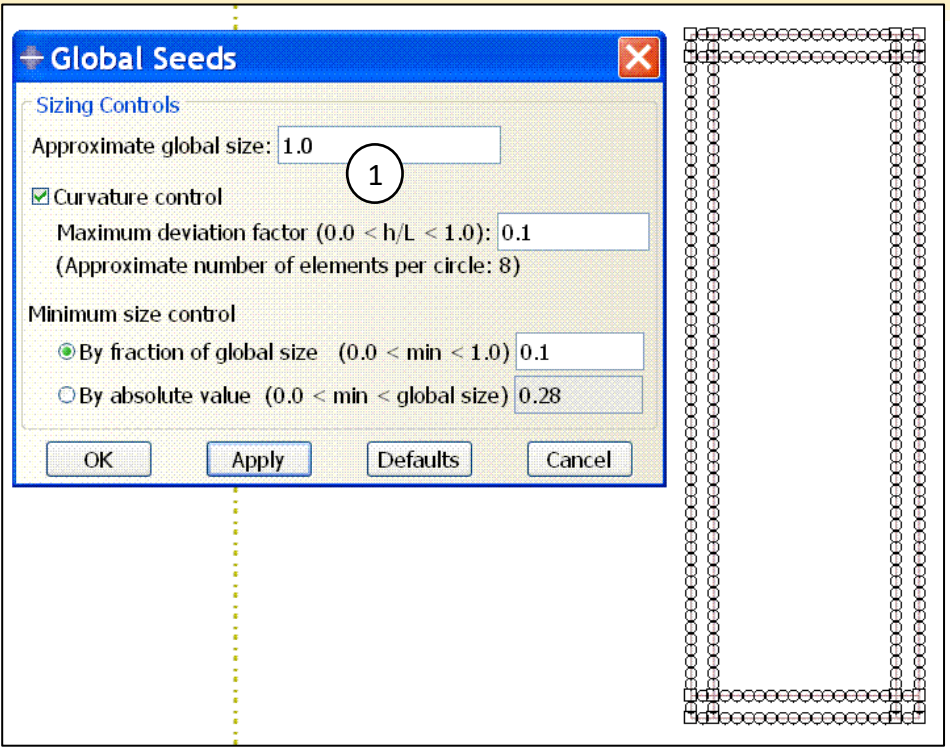




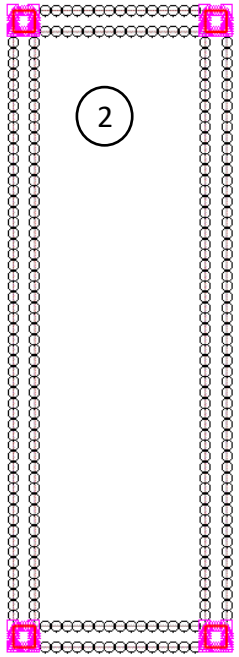
1. Select ***Create Lines: Connected*** to extend the partition lines in each corner to create squares and click the red ***X*** when complete
    - When creating a line, the “x”, “H” or “V” means the line is perfectly horizontal or vertical as seen in the images to the left
2. Click ***Done*** when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

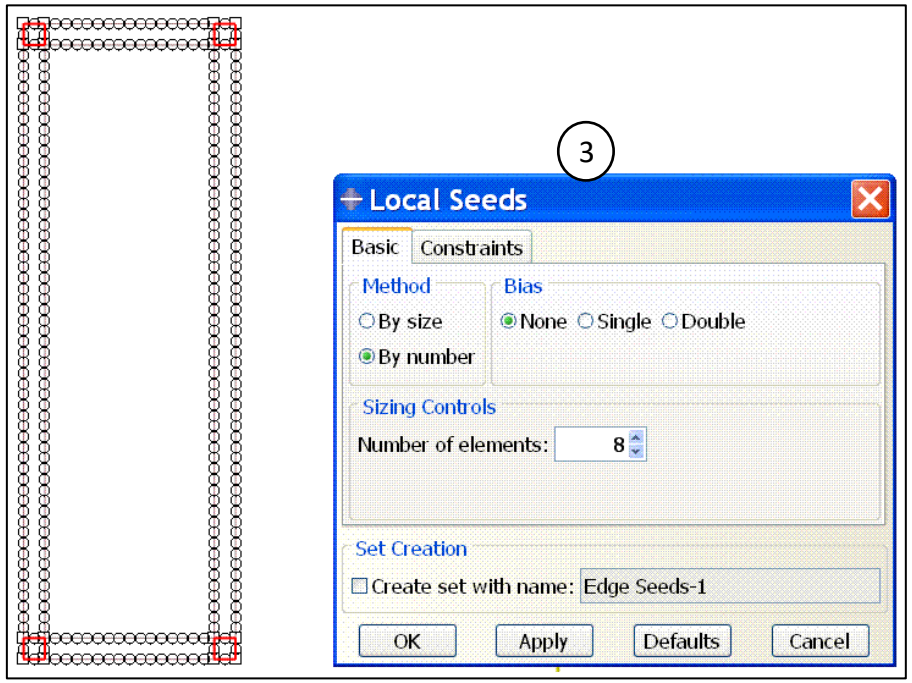


Step 4c: Meshing the Part

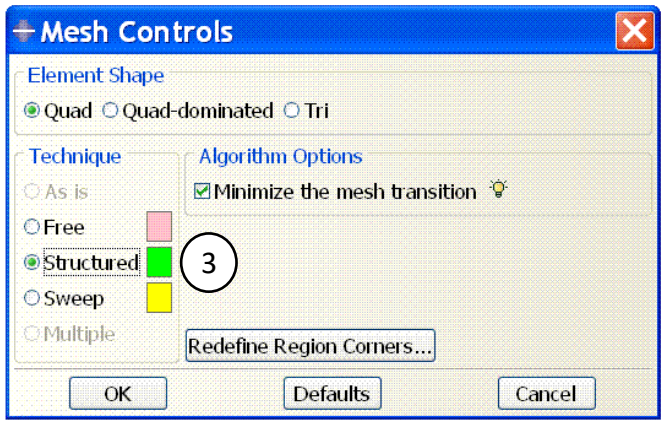


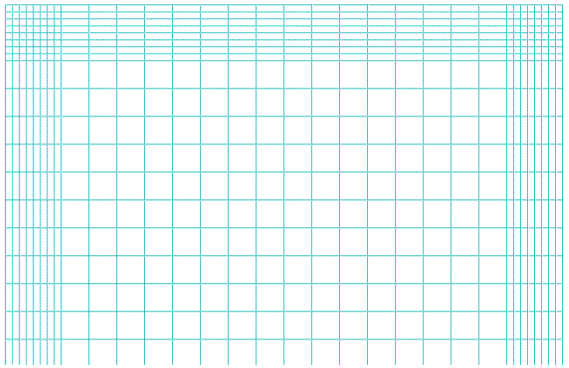
1. In the pulldown menu, select ***Seed → Part*** then set the ***Approximate global size*** to ***1.0*** and click **OK**
2. In the pulldown menu, now select ***Seed → Edges*** then select each edge of the four squares created during partitioning using ***Shift + Click***, then click ***Done***
3. In ***Method***, set the option to ***By Number*** and select ***8*** for ***Number of elements*** then click ***OK***

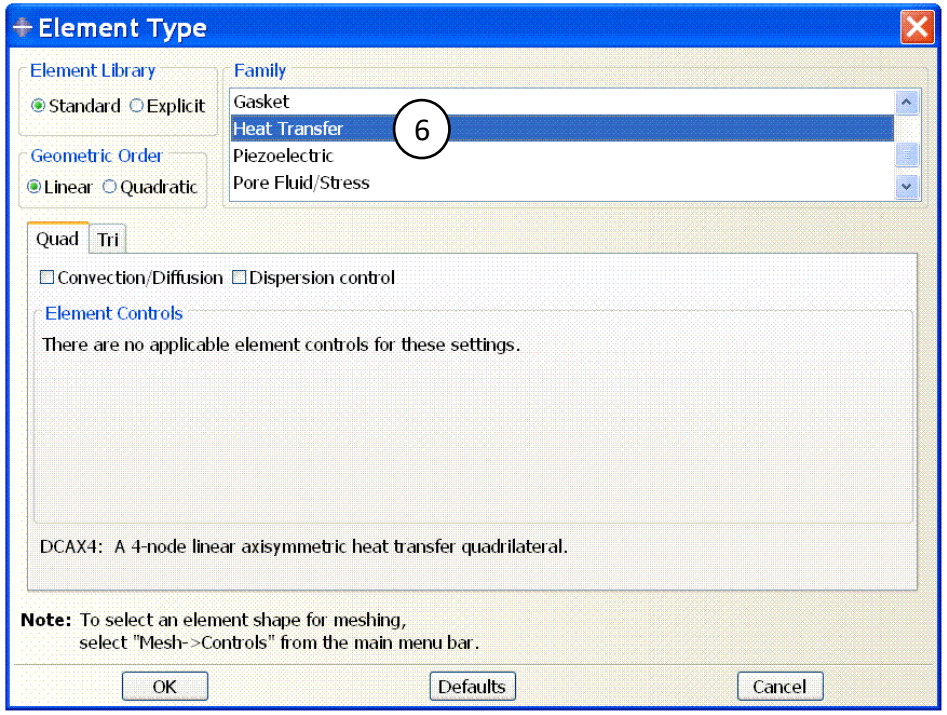




Step 4d: Meshing the Part

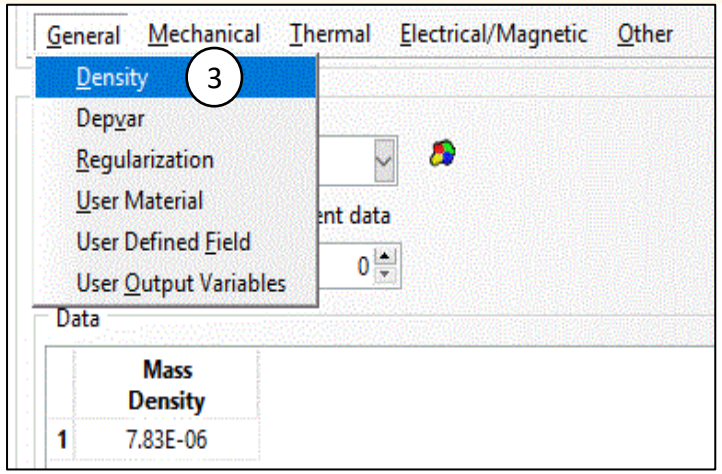


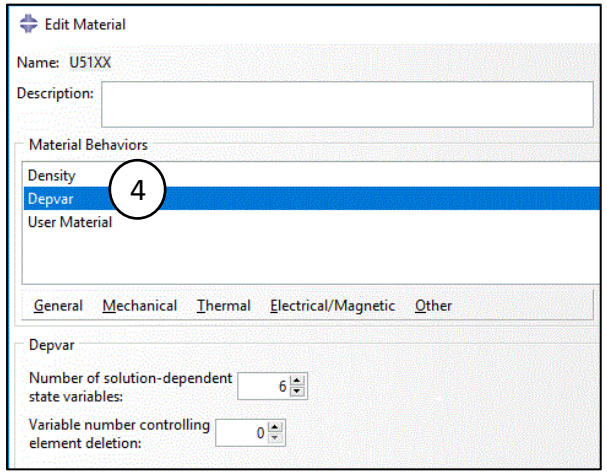


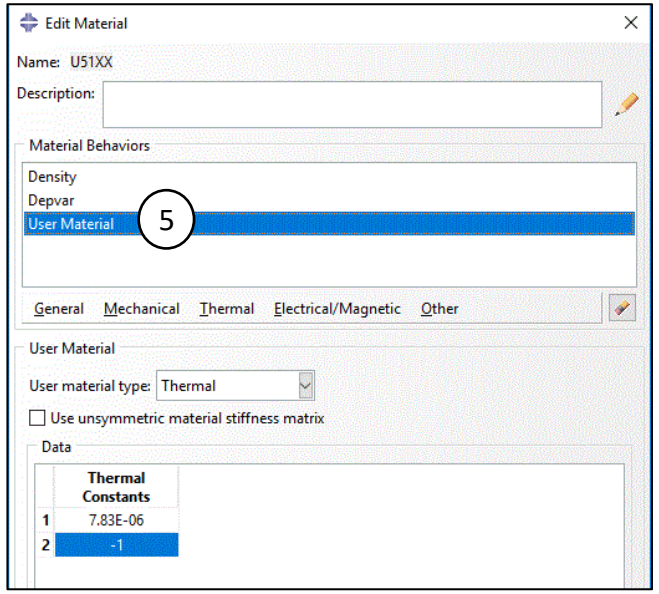


1. In the pulldown menu, select ***Mesh → Controls***
2. Box select the entire geometry then click ***Done***
3. Select ***Quad*** under ***Element Shape*** then select ***Structured*** under ***Technique*** and click ***OK*** when complete
4. In the pulldown menu, select ***Mesh → Element Type***
5.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

Step 5: Creating Material Properties

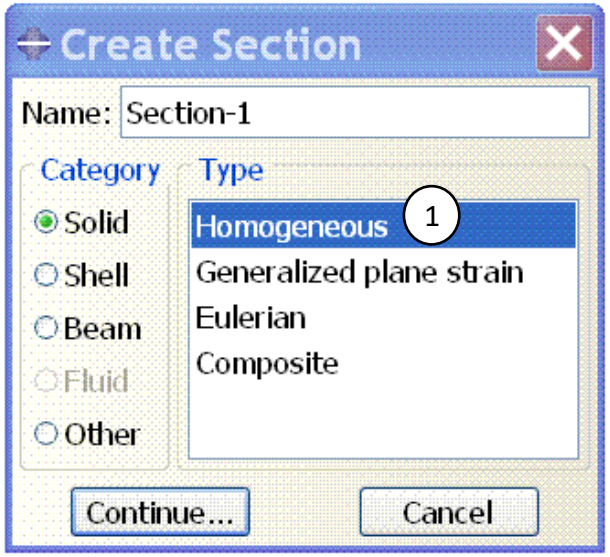


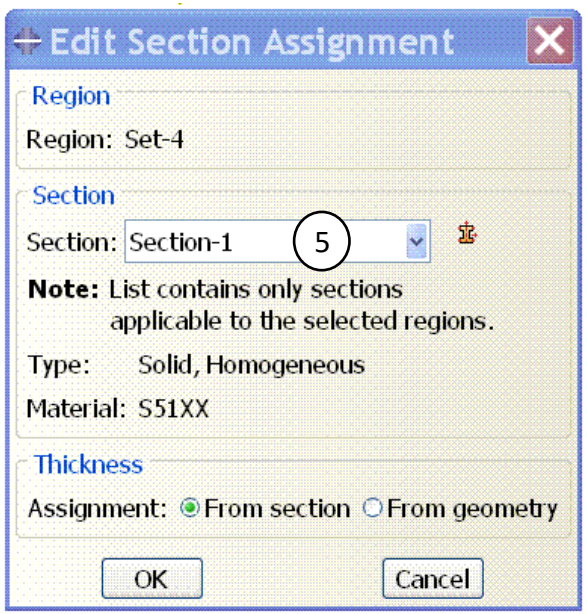


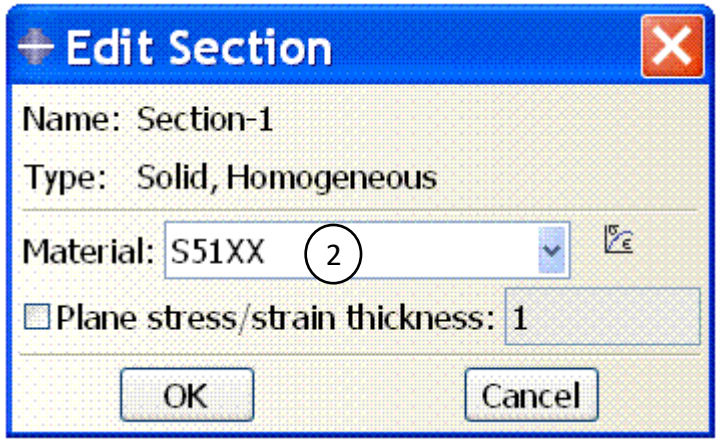


1. Under ***Module***, select ***Property*** to define material properties
2. In the pulldown menu, select ***Material → Create*** and name the material ***S51XX*** which is a specified material name in Dante Database
3. Define the density by selecting ***General → Density*** and typing in ***7.83E-06*** in the first cell
4. Define the number of Dante solution variables by selecting ***General → Depvar*** and typing in ***6*** for the ***Number of solution-dependent state variables***
5. Select ***General → User Material*** and ***Thermal*** under ***User material type:***
6. In the first cell, type in ***7.83E-06,*** hit ***Enter*** on the keyboard, and type in ***-1*** in the second cell to call the carburization subroutines
7. Click ***OK*** to exit the window

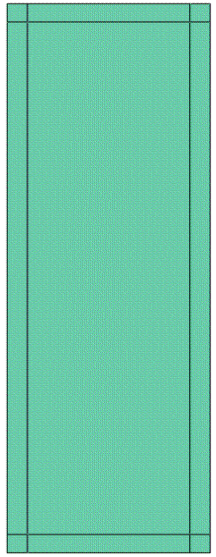
Step 5b: Creating Material Properties



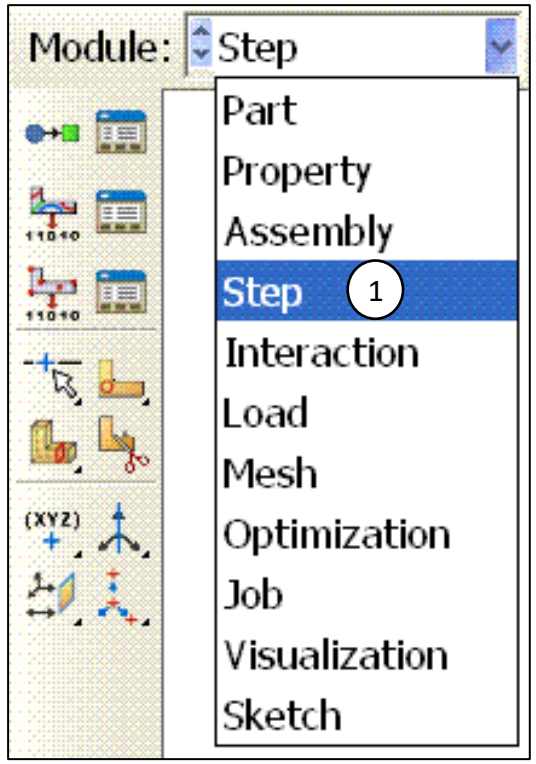


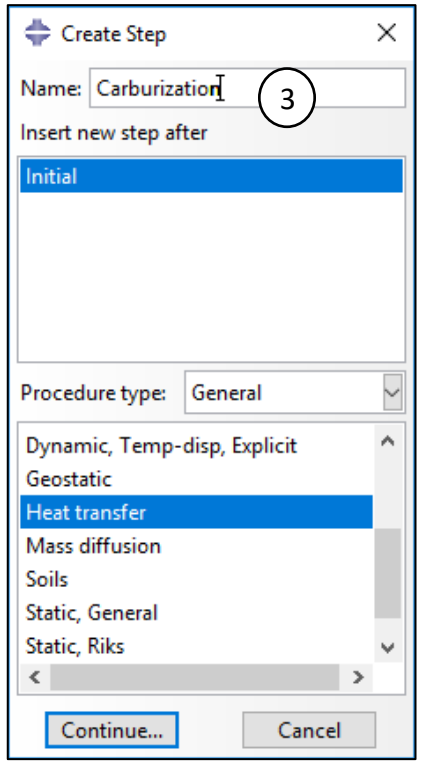


1. In the pulldown menu, select ***Section*** → ***Create*** then select ***Solid*** and ***Homogenous*** and click ***Continue***
2. Select the material ***S51XX*** and click ***OK*** to exit
3. To assign the section to the part, select ***Assign*** → ***Section*** in the pulldown menu
4. Box select the entire part then click done
5. Select the section to be assigned, then click ***OK***

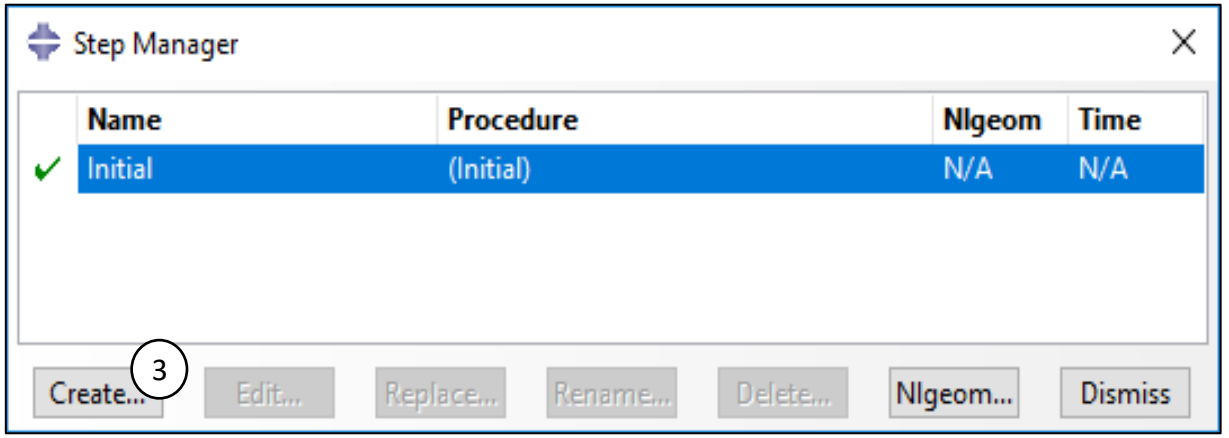


Step 6: Creating Simulation Steps

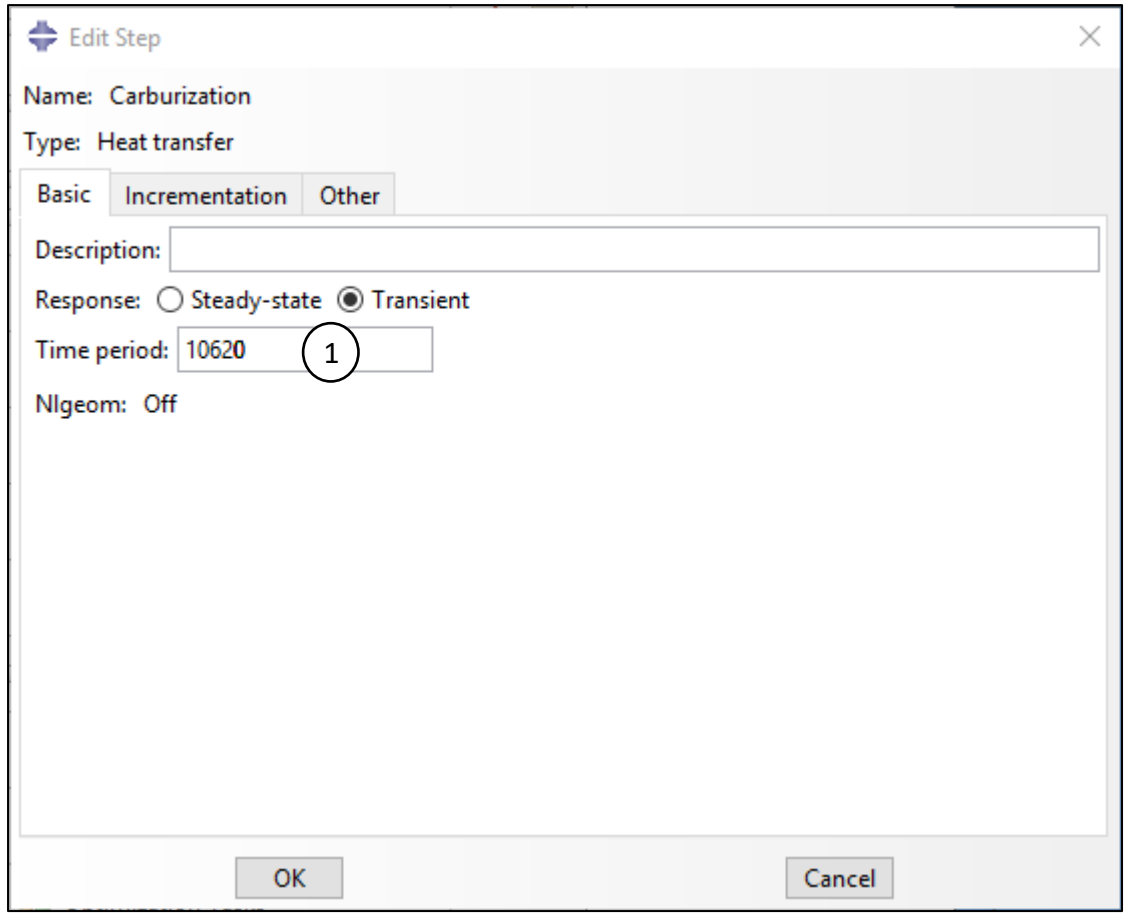




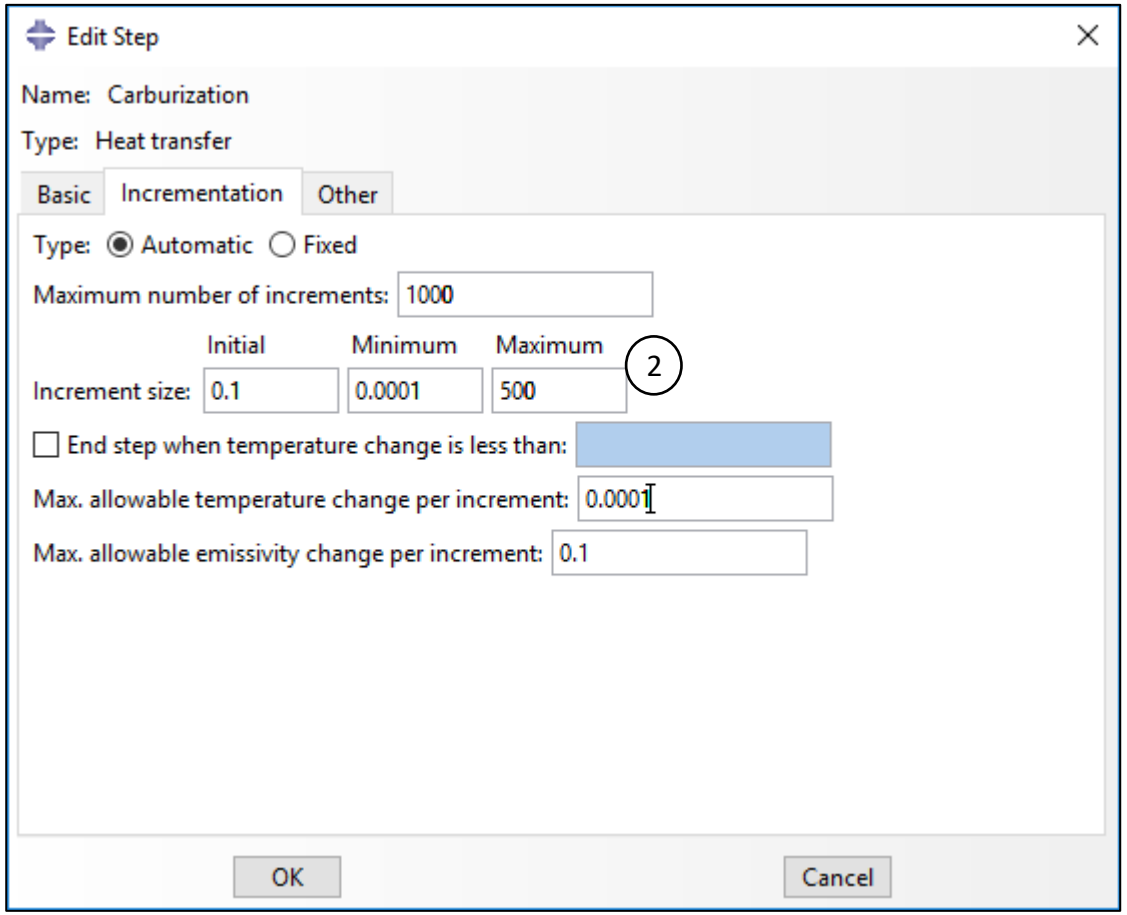
1. Under ***Module***, select ***Step*** to define the carburizing steps
2. In the pulldown menu, select ***Step*** → ***Manager*** and delete any previous steps excluding the ***Initial Step***
3. Select ***Create***, name the step an arbitrary name (***Carburization***), select ***Heat transfer*** under ***Procedure type:***, then click ***Continue***



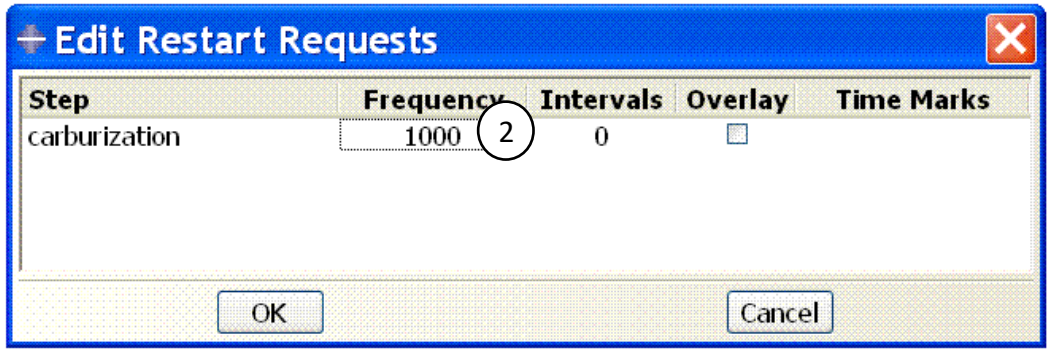
Step 6b: Creating Simulation Steps

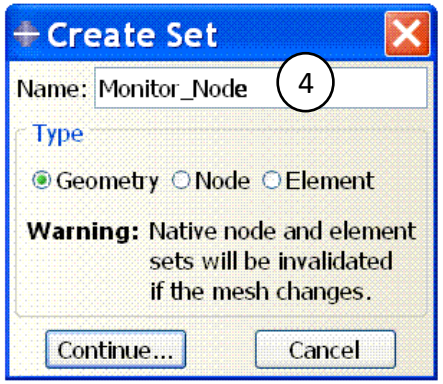


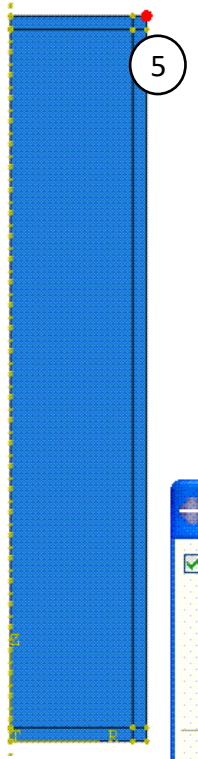
1. Set the ***Time period*** to ***10620*** (seconds) then click on the ***Incrementation*** tab
2. Define the ***Incrementation*** according to the figure, then click ***OK***



Step 7: Define Output Requests

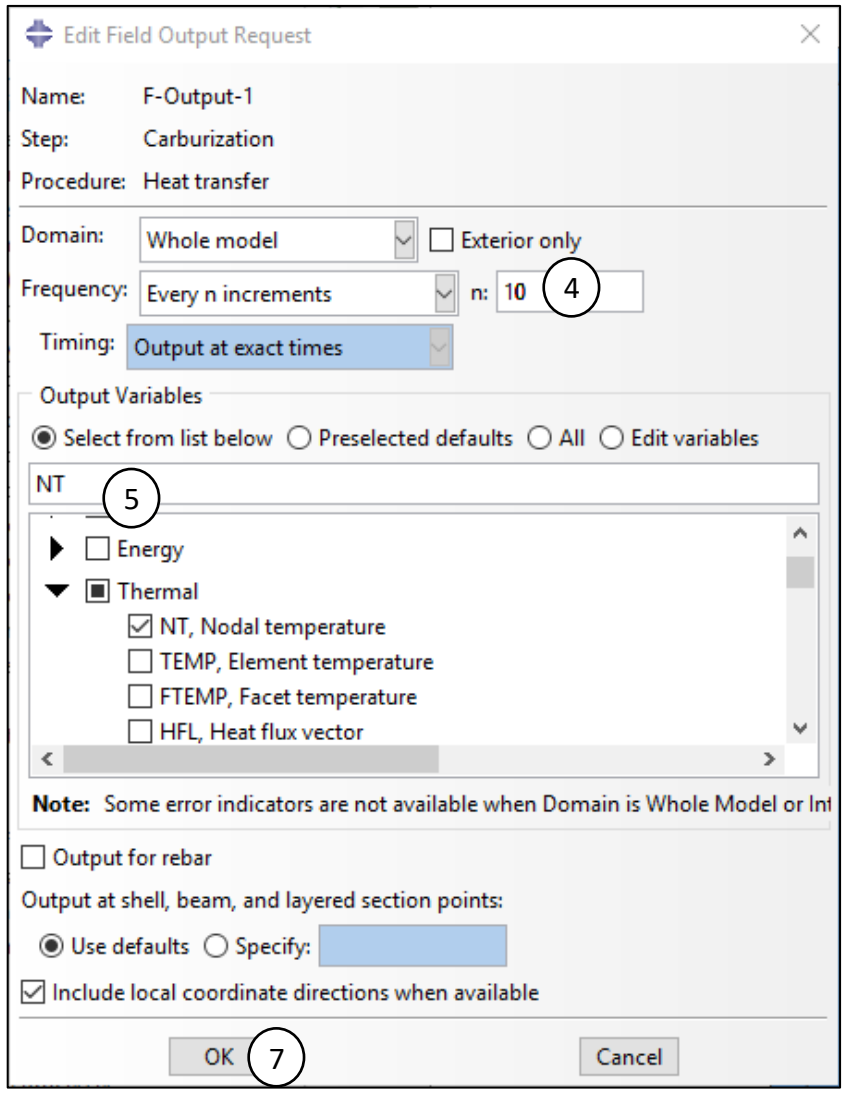






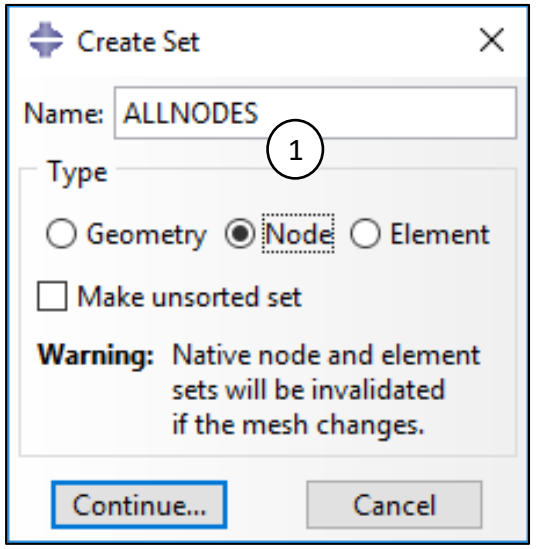
1. In the pulldown menu, select ***Output → Restart Requests***
2. Change the ***Frequency*** to ***1000*** for the carburization step then click ***OK***
3. In the pulldown menu, select ***Tools → Set → Create*** to define a node to monitor
4. Name the node ***Monitor_Node***, keep the default ***Geometry*** checked, and click ***Continue***
5. Select an arbitrary node (The node in the upper right corner) to be the ***Monitor_Node*** then click ***Done***
6. In the pulldown menu, select ***Output → DOF Monitor***, check the box that says ***Monitor a degree of freedom…***, select ***Monitor Node*** under region, and change ***Degree of freedom*** to ***11*** then click ***OK***

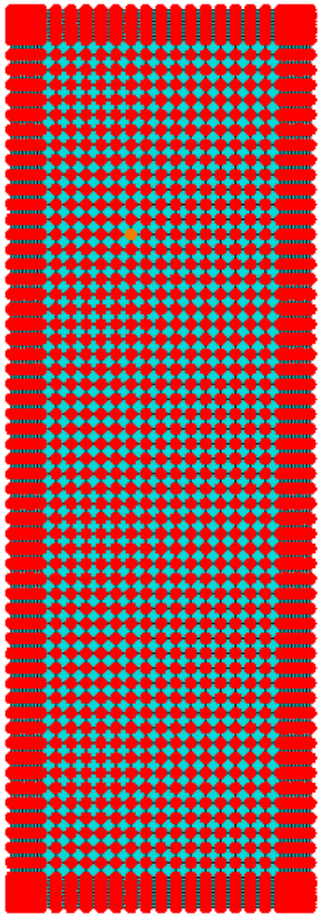
Step 7b: Define Output Requests



1. In the pulldown menu, select ***Output → Field Output Requests → Manager***
2. Delete all previous ***Field Output Requests*** and select ***Create***
3. Leave default settings and click ***Continue***
4. Change the ***Frequency "n"*** to ***10***
5. In ***Output Variables*** under ***Thermal***, select ***NT, Nodal temperature***
6. The rest can be set as default
7. Click ***OK*** to exit

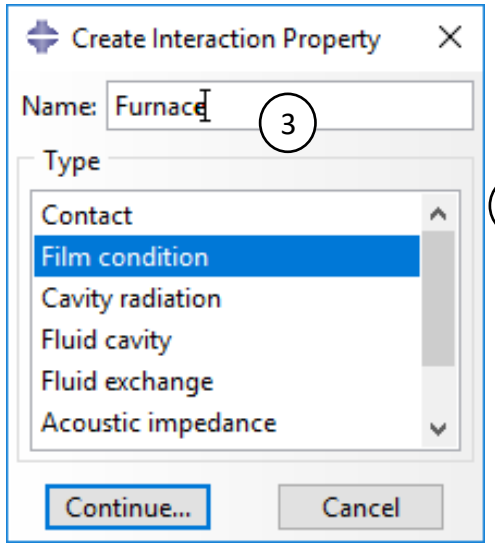
Step 7c: Defining Output Requests

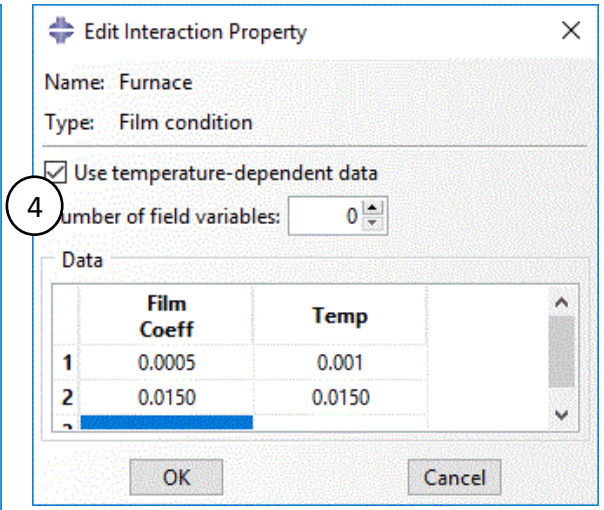


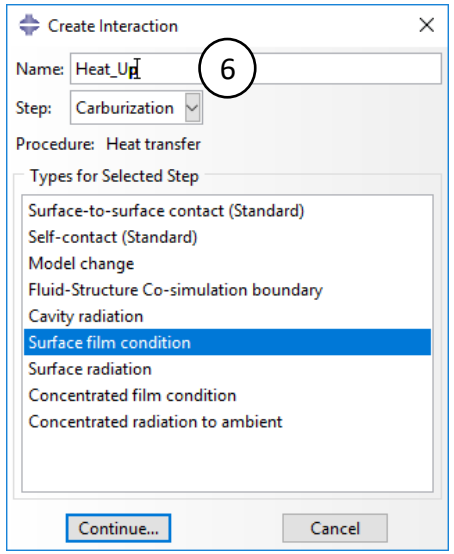


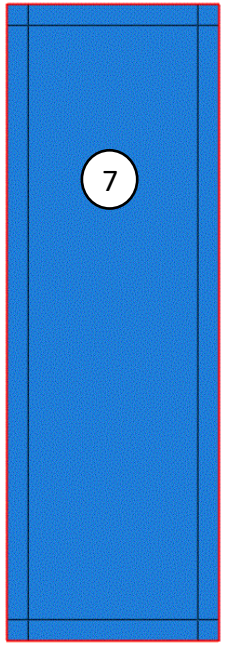
1. In the pulldown menu, select ***Tools*** → ***Set*** → ***Create***, change the name to ***ALLNODES***, and check the ***Node*** box then click ***Continue***
2. Box select the entire region then click done
3. This step will be used to define the initial conditions when editing the input file

Step 8: Define Boundary Conditions



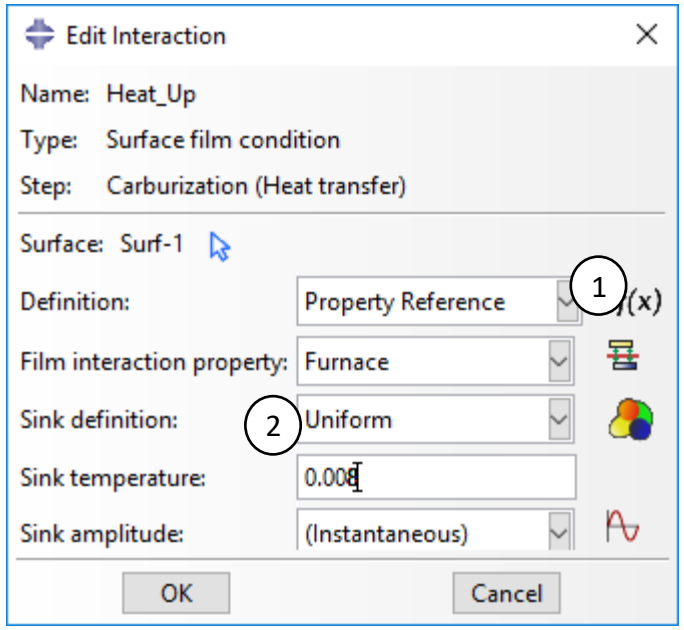




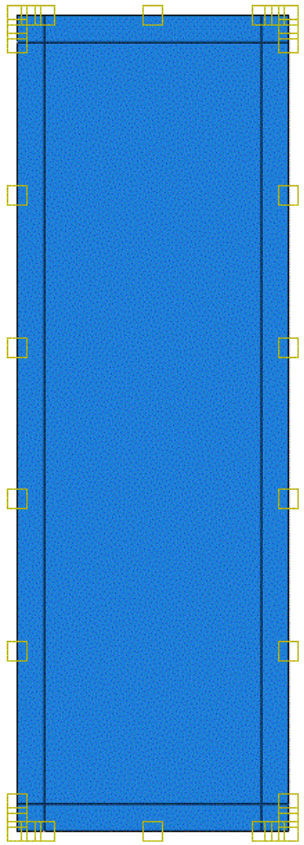


1. Under ***Module***, select ***Interaction*** to define the surface film conditions
2. In the pulldown menu, select ***Interaction → Property → Create***
3. Name an arbitrary name such as ***Furnace***, select ***Film condition***, then click ***Continue***
4. Check the box that says, ***Use temperature-dependent data*** and type in the corresponding film coefficients as seen in the image to the left, then click ***OK***
   - This can also be edited in the input file
5. Then select ***Interaction → Create***
6. Name an arbitrary name, select ***Carburization*** for the ***Step***, select ***Surface film condition***, then click ***Continue***
7. Select the surface(s) to be carburized then click ***Done***
   - Box select the entire part in this case

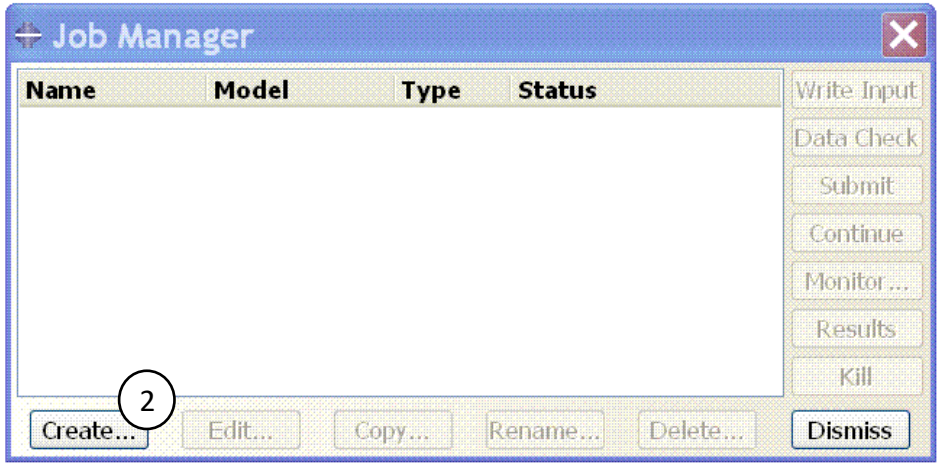
Step 8b: Define Boundary Conditions

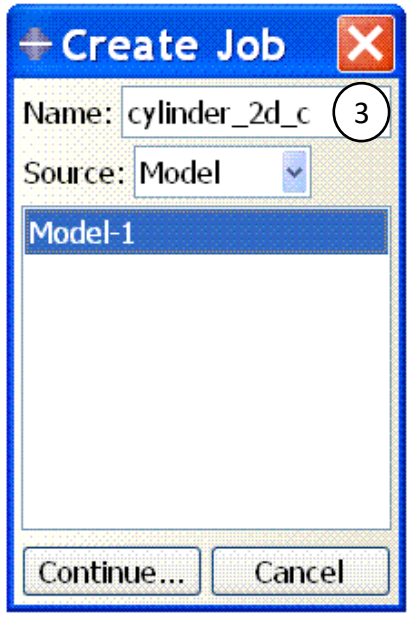


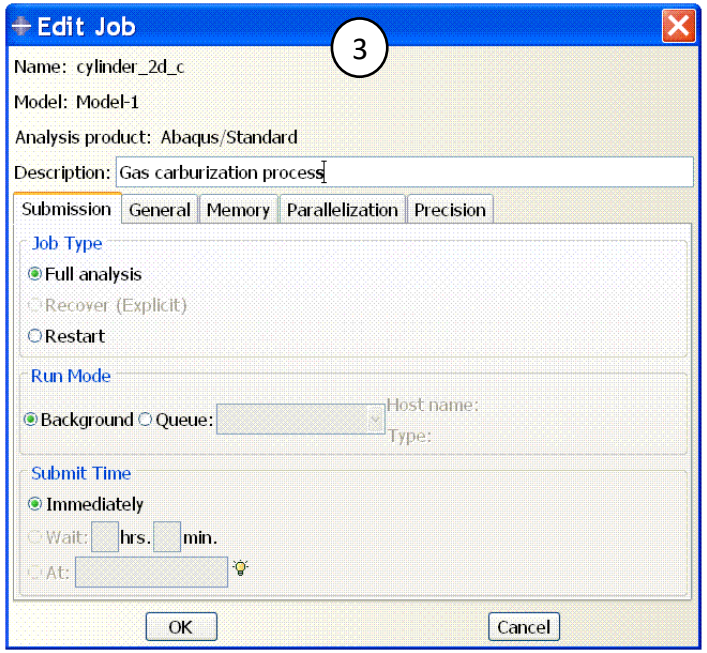
1. In the ***Edit Interaction*** window, select ***Property Reference*** under the ***Definition*** tab
2. Select ***Furnace*** under the ***Film interaction property:*** tab and set the ***Sink Temperature*** to ***0.008***, then click ***OK***



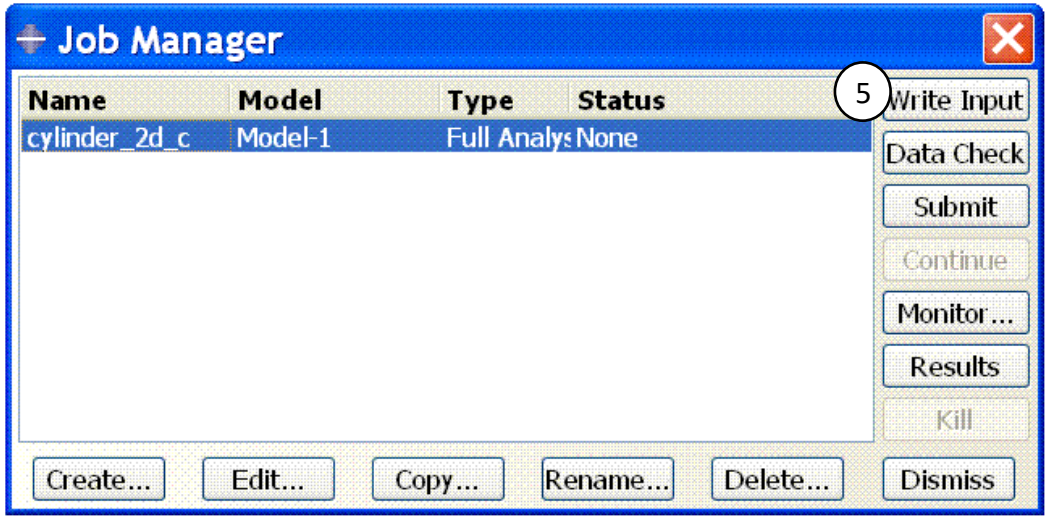
Step 9: Write Out Input Deck







1. Under ***Module***, select ***Job***
2. In the pulldown menu, select ***Job → Manager*** and click ***Create***
3. Give the job a name such as ***simple_2d_ring_c*** then click ***Continue***
4. Use the default values given in ***Edit Job*** and click ***OK***
5. Go back to ***Job Manager*** and click ***Write Input*** to write out the input file to the working directory



Step 10: Modifying The Input File

```
4766 ** MATERIALS
4767 **
4768 *Material, name=S51XX
4769 *Density
4770  7.83e-06,
4771 *Depvar
4772       6,
4773       1, Temperature,  Carburing Temperature
4774       2, CRBALL,       Carbon All
4775       3, CRBDA,        Weight Fraction of Carbon in Carbide A Form
4776       4, CBDASIZE,     Carbide A Size Factor
4777       5, CRBDB,        Weight Fraction of Carbon in Carbide B Form
4778       6, CBDBSIZE,     Carbide B Size Factor
```

2

1. Open the .inp file found in the working directory to begin editing
2. Under the ***MATERIALS*** section near the bottom of the file, paste
   1, Temperature, Carburizing Temperature
   2, CRBALL, Carbon All
   3, CRBDA, Weight Fraction of Carbon in Carbide A Form
   4, CBDASIZE, Carbide A Size Factor
   5, CRBDB, Weight Fraction of Carbon in Carbide B Form
   6, CBDBSIZE, Carbide B Size Factor
3. Under the ***INTERACTION PROPERTIES*** section, paste
   *INITIAL CONDITIONS, TYPE=FIELD, VAR=1
   ALLNODES, 900.0
   *Initial Conditions, type=TEMPERATURE
   ALLNODES, 0.002
4. Under the ***STEP*** section, type in ***amp=step*** and ***op=new*** in the appropriate place as shown in the image

```
4782 ** INTERACTION PROPERTIES
4783 **
4784 *Film Property, name=Reaction_Factor
4785 0.0005, 0.001
4786 0.015, 0.015
4787 **
4788 ** PREDEFINED FIELDS
4789 **
4790 ** Name: CarburizingTemperature-1   Type: Field
4791 *Initial Conditions, type=FIELD, variable=1
4792 ALLNODES, 900.
4793 ** Name: Total_CarbonWt-1   Type: Temperature
4794 *Initial Conditions, type=TEMPERATURE
4795 ALLNODES, 0.002
```

3

```
4790 ** STEP: Carburization
4791 **
4792 *Step, name=Carburization, nlgeom=NO, inc=1000, amp=step
4793 *Heat Transfer, end=PERIOD, deltmx=0.0001
4794 0.1, 10620., 0.0001, 500.,
4795 **
```

4

```
4796 ** INTERACTIONS
4797 **
4798 ** Interaction: Heat_Up
4799 *Sfilm, op=new
4800 Surf-1, F, 900., Furnace
4801 **
```

4

Step 10b: Modifying The Input File

```
7546 ** ----------------------------------------------
7547 ** carburization Stage
7548 **
7549 *Step, name=carburize1, inc=1000, AMP=STEP
7550  carburization process 1
7551 *Heat Transfer, end=PERIOD, deltmx=0.0005
7552 0.1,  28800.,0.0000001,  500.
7553 **
7554 *CONTROLS, PARAMETERS=LINE SEARCH
7555 6,
7556 *CONTROLS, PARAMETERS=TIME INCREMENTATION
7557 20, 30
7558 *CONTROLS, FIELD=TEMPERATURE, PARAMETERS=FIELD
7559 0.005, 0.005
7560 **
7561 *Sfilm, OP=NEW
7562  Exposed_Surface, F, 0.0095, Reaction_Factor
7563 **
```

1

```
4802 ** OUTPUT REQUESTS
4803 **
4804 *Restart, write, frequency=1000
4805 *Monitor, dof=11, node=Monitor_Node, frequency=1
4806 **
4807 ** FIELD OUTPUT: F-Output-1
4808 **
4809 *Output, field, frequency=10
4810 *Node Output
4811 NT,
4812 *Element Output
4813 SDV1,SDV2,SDV3,SDV4,SDV5,SDV6
4814 *Output, history, frequency=0
4815 *End Step
```

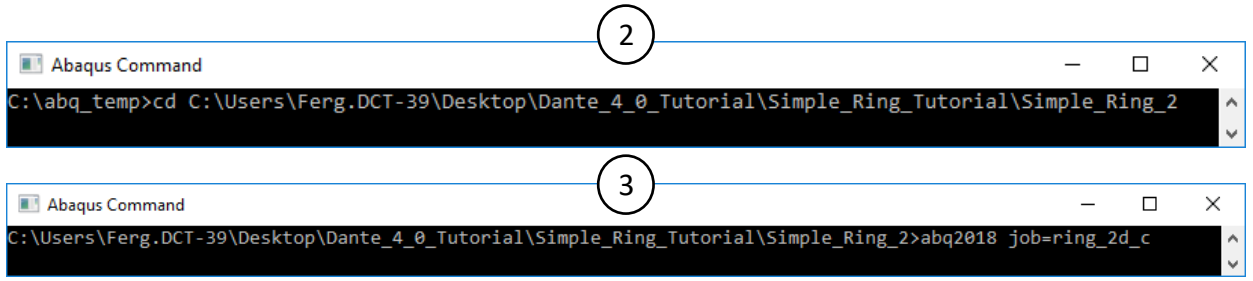
2

1. Under the ***Step*** section, add
   *CONTROLS, PARAMETERS=LINE SEARCH
   6,
   *CONTROLS, PARAMETERS=TIME INCREMENTATION
   20, 30
   *CONTROLS, FIELD=TEMPERATURE,
   PARAMETERS=FIELD
   0.005, 0.005

2. Under the ***Step*** section, add the following directly below the ****Node Output*** definition
   *Element Output
   SDV1,SDV2,SDV3,SDV4,SDV5,SDV6

3. Before moving on to the next step, please be sure there are ***no blank lines*** in the input file. Failure to do so may cause the model to not run.

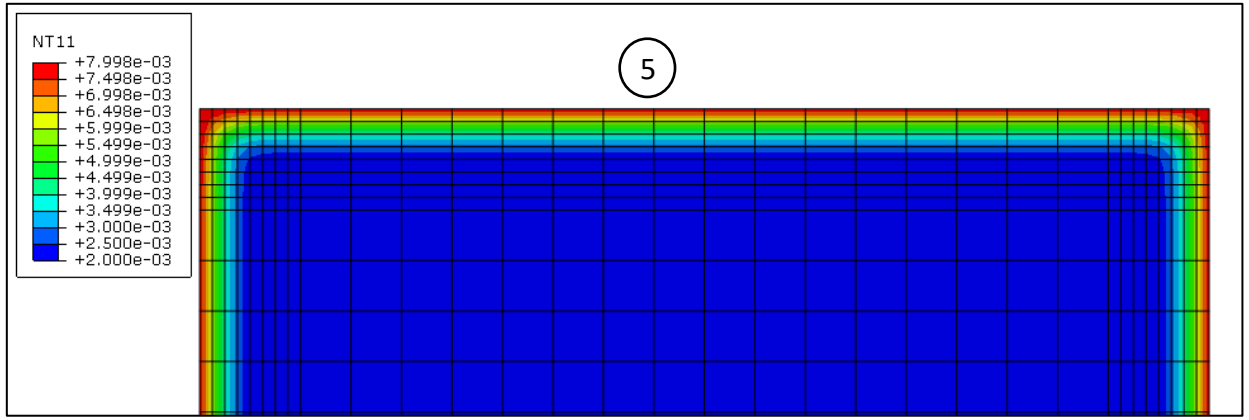
Step 11: Running the Job/Verifying the Results



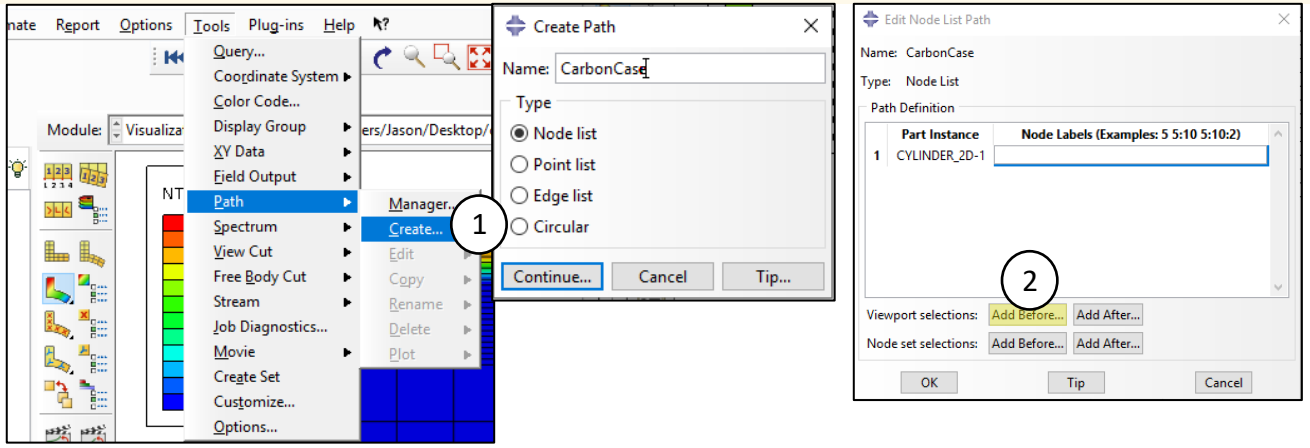
1. Save the input and open a new ***Abaqus Command*** window
2. Change the directory to where the input file is located using the command "***cd InputFileLocation"*** and press enter
3. Then use the command ***"abqxxxx job=InputFileName"*** to run the job
   - xxxx represents the version of Abaqus
4. To open the file, go back to ***Abaqus Cae***, under ***Module***, select ***Visualization*** and open the .odb file located in the working directory

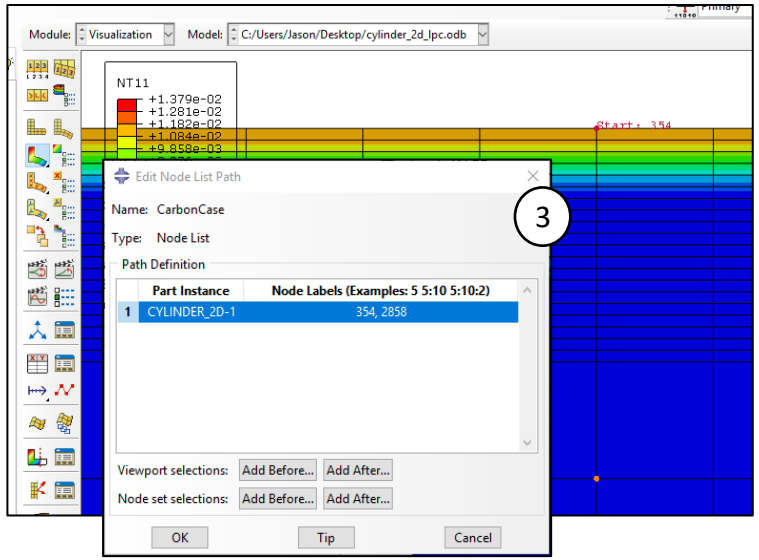
5. With the part open in the ***Visualization*** module, check the ***NT11*** Abaqus variable to see the free carbon contour.
6. As with all simulations, take a moment to check the result to ensure that the results make sense. We expect higher carbon in the corners and a notable carbon gradient from surface to core.
7. In the ***Field Output*** menu, one can select several DANTE solution dependent variables to display on the contour. While the Abaqus NT11 variable represents free carbon, the ***SDV_CBDASIZE*** variable will show the relative size of carbide A, ***SDV_CRBDA*** shows the percent of carbon in carbide A, ***SDV_CRBALL*** will show all carbon in the part(free carbon and carbides), while ***SDV_CRBDBSIZE*** and ***SDV_CRBDB*** are reserved for size of carbon B and carbon in carbide B, which will soon be implemented.

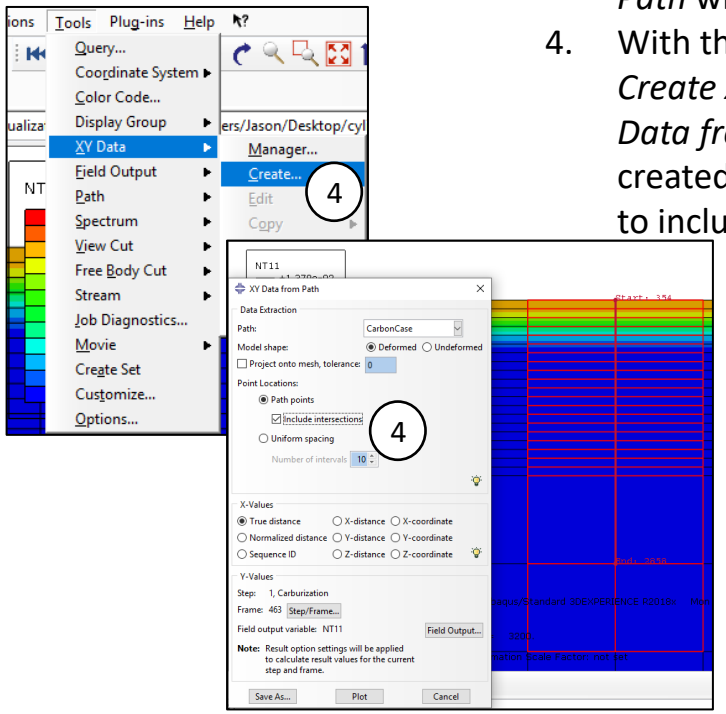
Note: Carbide size above a value of 5 becomes more stabile and less likely to decompose into free carbon.



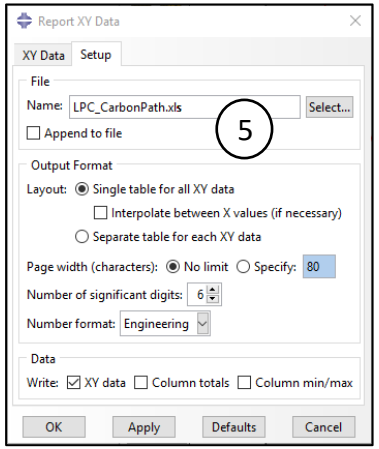
Step 12: Set up a Path Plot





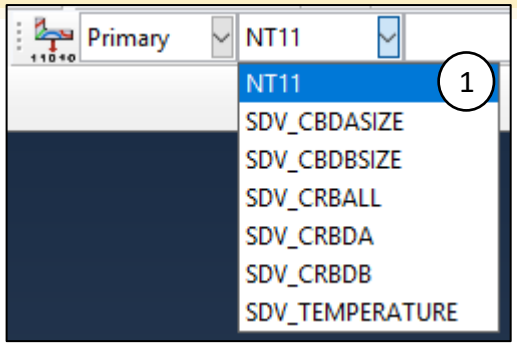


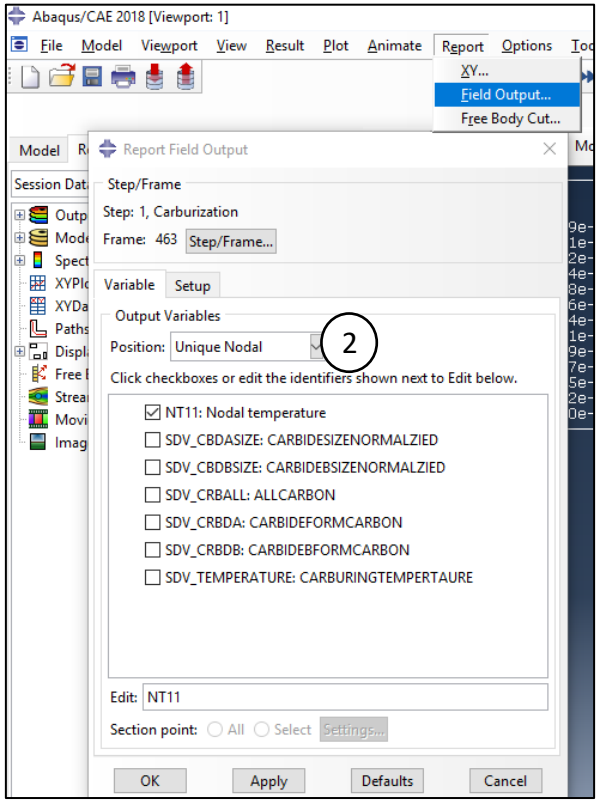
1. A path plot is desirable for carbon in order to easily see carbon case depths. While in the ***Visualization*** module, go to ***Tools → Path → Create*** to open the ***Create Path*** dialog window. Name the path and select a *Node list* and click continue to select the nodes to make the path.
2. 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 window, 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 Excel file.

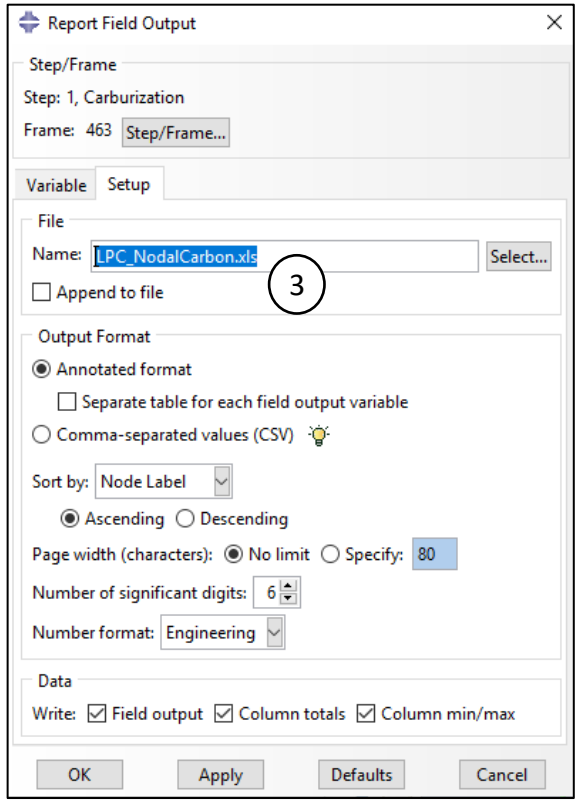


Note: See Step 13b, #1-2 for help with delimiting the data.

Step 13: Post Processing (Write out Carbon profile)



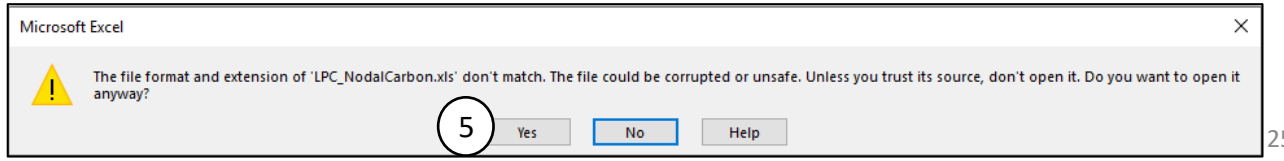




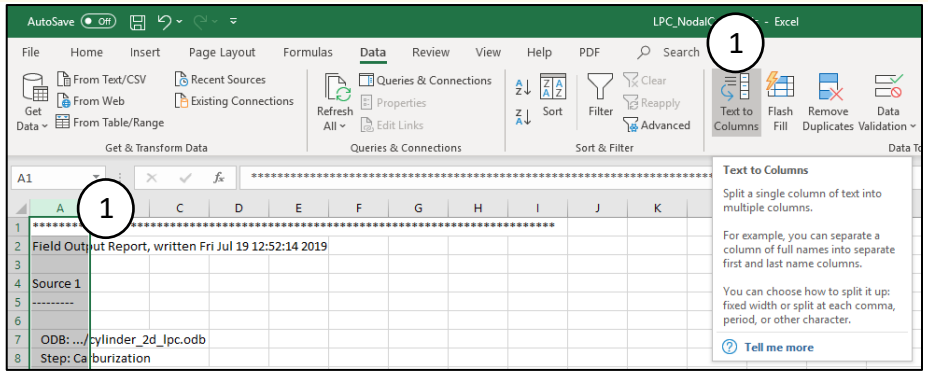
1. With the part open in the Abaqus Visualization module, select NT11 as the field to display
2. In the Abaqus menu, go to ***Report → Field Output***. In the ***Report Field Output*** window select ***Unique Nodal*** as ***Position:***, and check the box next to ***NT11:Nodal Temperature***
3. In the same window, on the setup tab, change the file extension to .xls in order to open the data in excel. Name the file and the location to save to as you like, then click ***OK*** to save the file

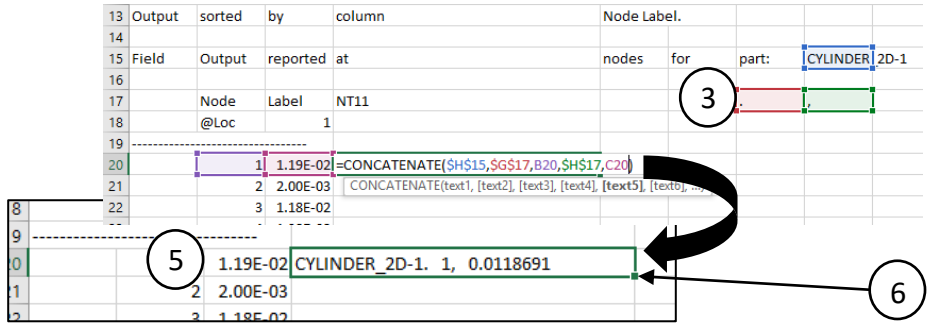
Note: uncheck Append to file to ensure data file is unique, and not appended to previous data.

4. Navigate to the directory where the carbon file is written and double click to open in Excel
5. Excel will display a corrupted or unsafe file warning, ignore this and select Yes to open the data file



Step 13b: Post Processing (Saving the Carbon data)

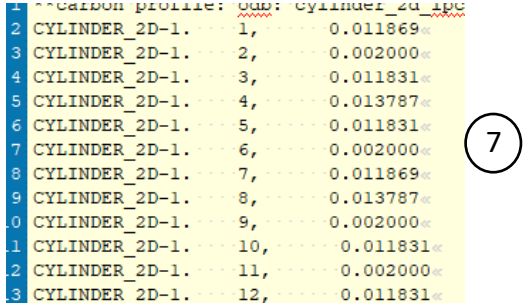




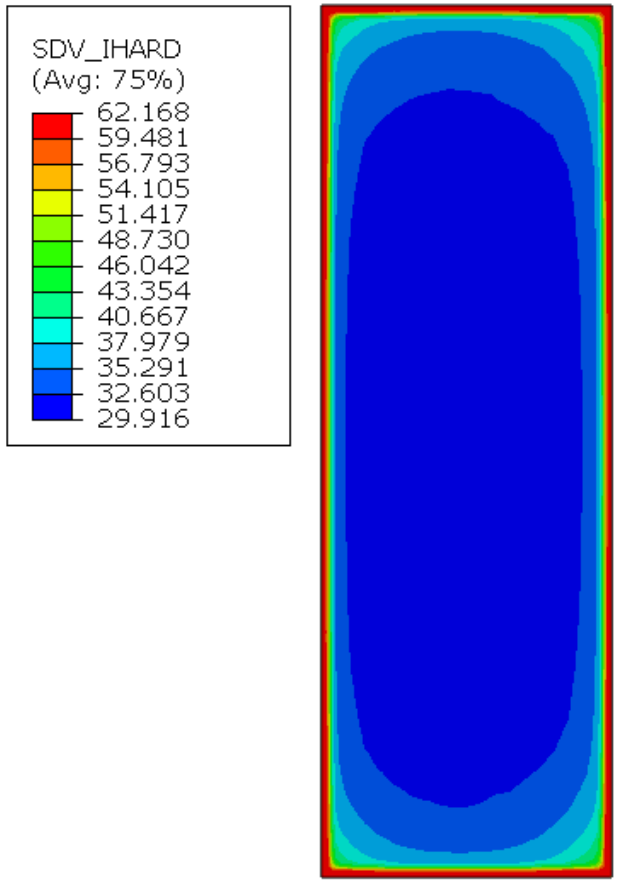
1. With the data open in excel, select the ***A column***, and under the ***Data*** menu click ***Text to Columns***
2. In the popup window, select ***Delimited*** and click ***Next***. In the following window choose to delimit by ***Space*** and click ***Finish***. This formats the data by cells in order to easily format the text
3. In any empty cell add a period followed by 5 or so spaces and, in another cell, add a comma followed by 5 or so spaces. We will use these to format the data for the Abaqus .nod file
4. Abaqus requires the nodal input file to be formatted as follows:

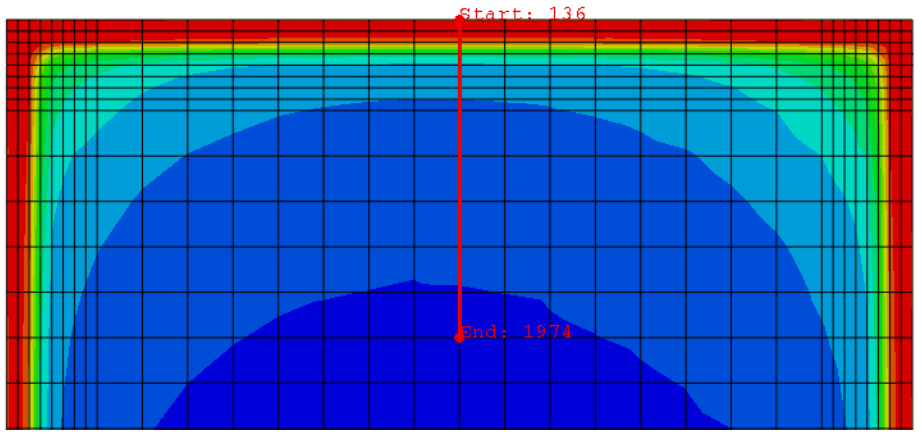
   PartName. NodeNumber, CarbonValue
5. Directly next to the first-nodal carbon value type "=CONCATENATE(" and select the appropriate cells to format the data. A sample is provided below.

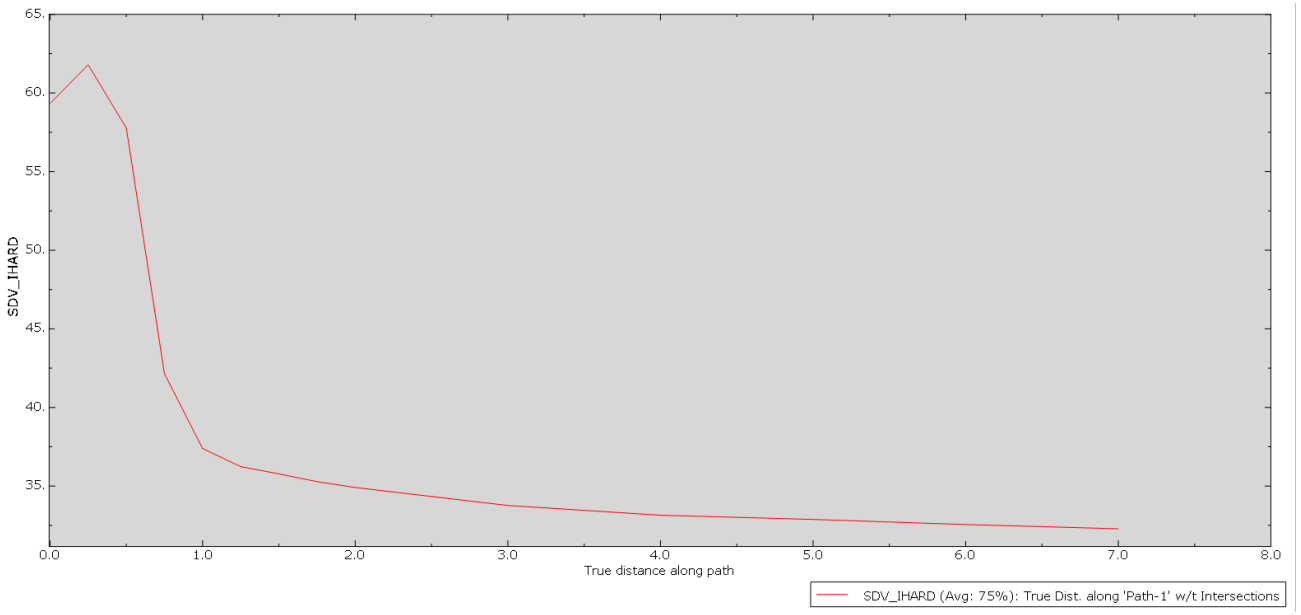
   Note: The $ is needed to keep the cell in position. (F4 is the shortcut that will add this character to the selected cell)
6. Double click on the box in the bottom-right of the cell that was just created to apply this format the rest of the carbon data
7. Copy and paste this formatted data into your favorite text editor and save as a filename with a .nod extension. (i.e.. "simple_ring_cc.nod")



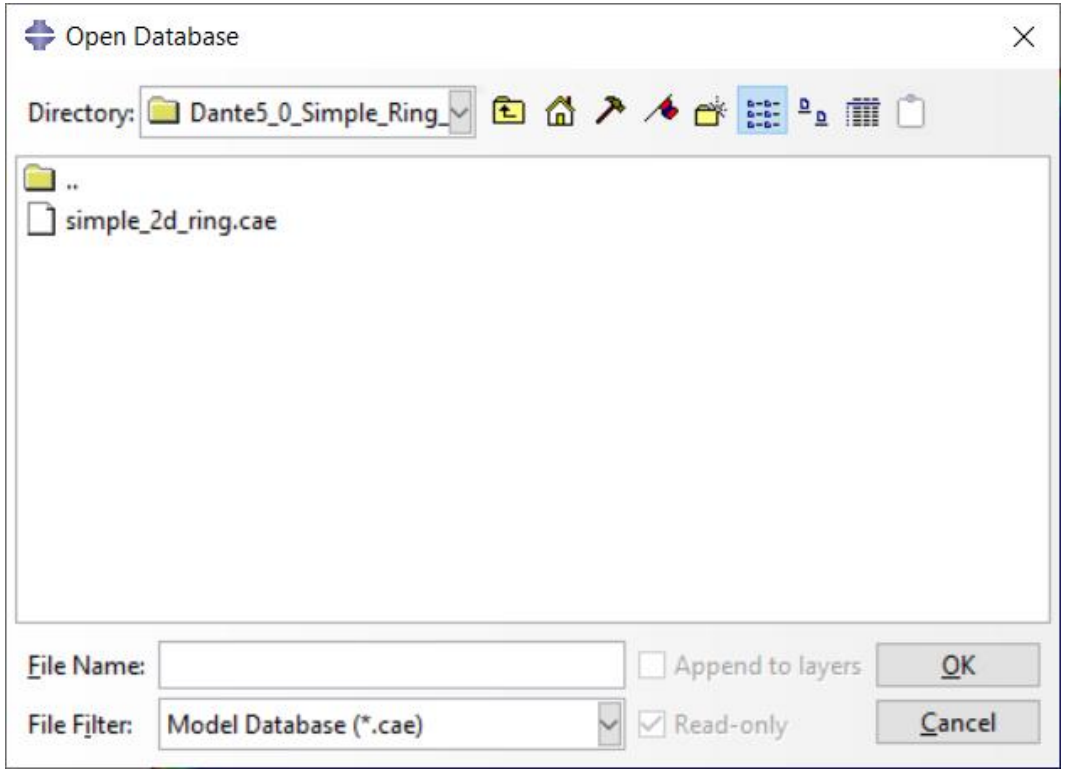
Thermal Model





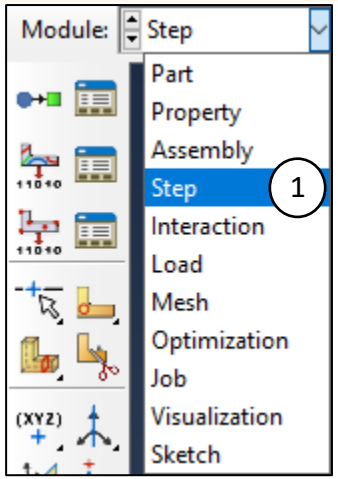


Step 1: Setting Up The CAE



1. Once the carbon content text file has been generated, go to ***Module*** and select ***Part***
2. Open the .cae file created for the carburization model if it is still not open
    - The cae carburization model will be edited to create the thermal model

Step 2: Defining The Steps



- Step 1: Heating up step

```
*Step, name=Heat-Up, amp=step, inc=1000
Furnace heat up
*Heat Transfer, end=PERIOD, deltmx=30.
1., 1000., 1e-06, 100.,
```

6

- Step 2: Carburization step

```
*Step, name=Carburization, amp=step, inc=1000
Carburization process
*Heat Transfer, end=PERIOD, deltmx=30.
1., 10620., 1e-06, 500.,
```

- Step 3: Air transfer step

```
*Step, name=Air-Transfer, amp=step, inc=1000
Transfer from furnace to quench tank
*Heat Transfer, end=PERIOD, deltmx=30.
0.01, 12., 1e-05, 1.,
```

- Step 4: Immersion quenching step

```
*Step, name=Immersion-Quench, amp=step, inc=1000
*Heat Transfer, end=PERIOD, deltmx=30.
0.001, 5., 1e-06, 1.,
```

- Step 5: Oil quenching step

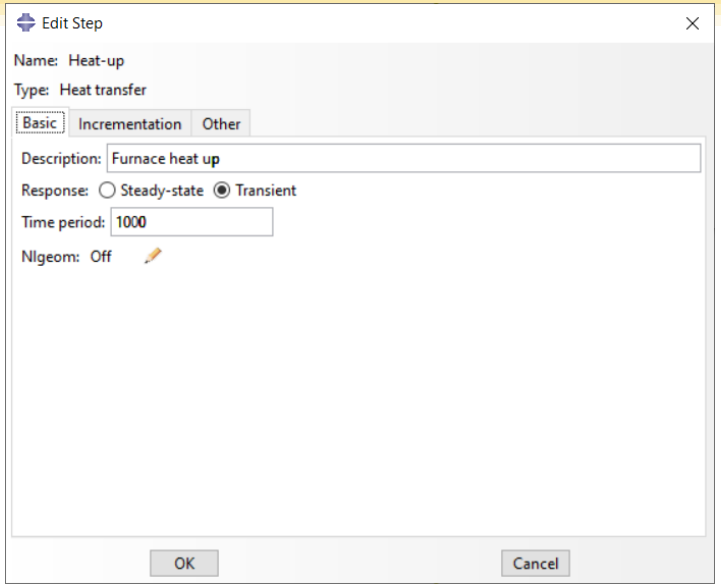
```
*Step, name=Oil-Quench, amp=step, inc=1000
Oil quench to 65 C
*Heat Transfer, end=PERIOD, deltmx=30.
0.001, 1000., 1e-05, 50.,
```

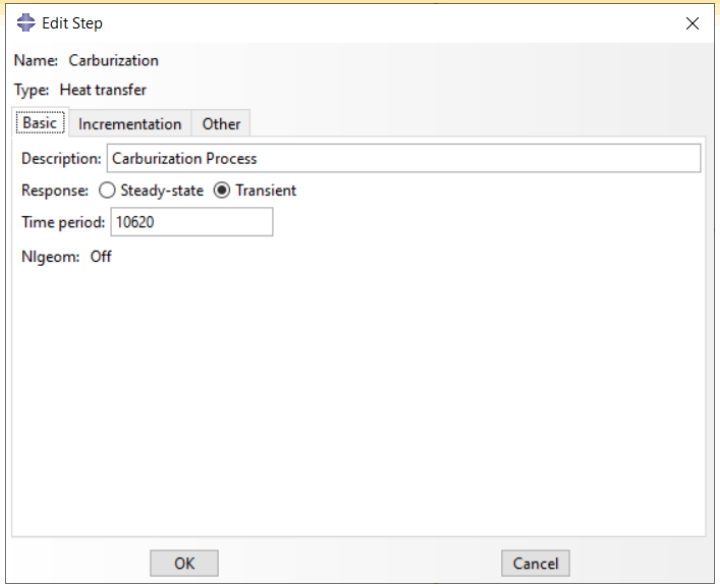
- Step 6: Air cooling to room temperature

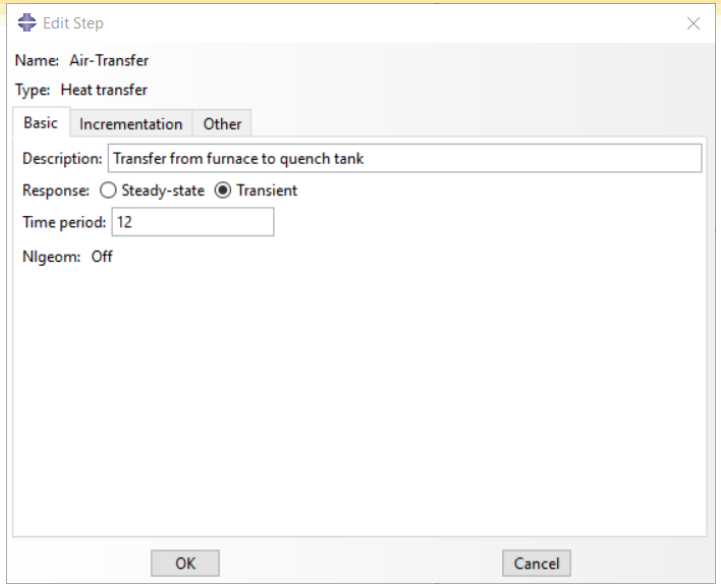
```
*Step, name=Air-Cool, amp=step, inc=1000
Air cool to room temperature
*Heat Transfer, end=PERIOD, deltmx=30.
1., 1500., 1e-05, 100.,
```

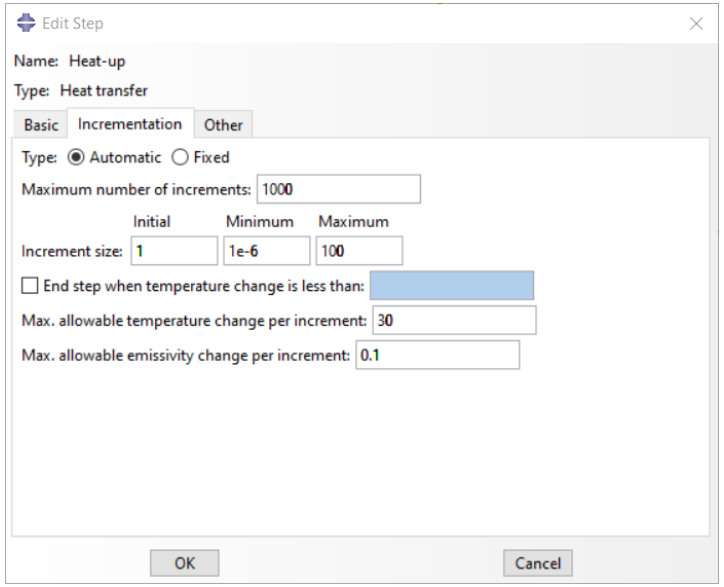
1. Under ***Module***, select ***Step*** to define the heat transfer analysis steps
2. In the pulldown menu, select ***Step*** → ***Manager*** and delete the ***Carburization*** step
3. A ***Heat-Up***, ***Carburization***, ***Air-Transfer***, ***Immersion-Quench***, ***Oil-Quench,*** as well as an ***Air-Cool*** step will be created
4. In ***Step Manager***, click ***Create***
5. Name the first step ***Heat-Up***, select ***Heat Transfer***, and click ***Continue***
6. The same procedure will be used for each step with the only difference being the ***Name of the Step***, ***Time Period***, and ***Increment Size***
   - Values such as ***Max Increment*** and ***Max allowable temperature/emissivity change*** will stay the same in this instance
   - The names and values for the 6 steps are shown on the following 2 slides as entered in the ***Edit Step*** boxes

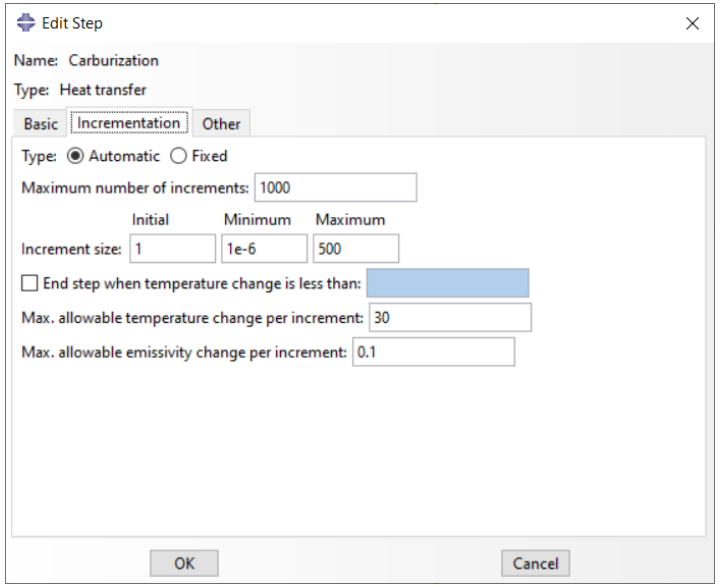
NOTE: If step 4: Immersion quenching step is not desired, do not add the step and film quench text file in Step 9. The Dante Model Builder plugin tutorial does not include this step. If desired, follow that tutorial, write the input file from the plugin and then follow these steps to add immersion quench.

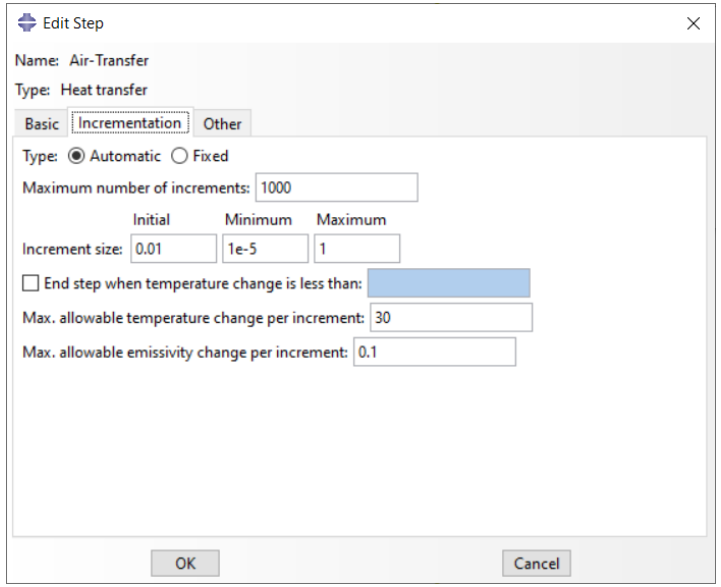


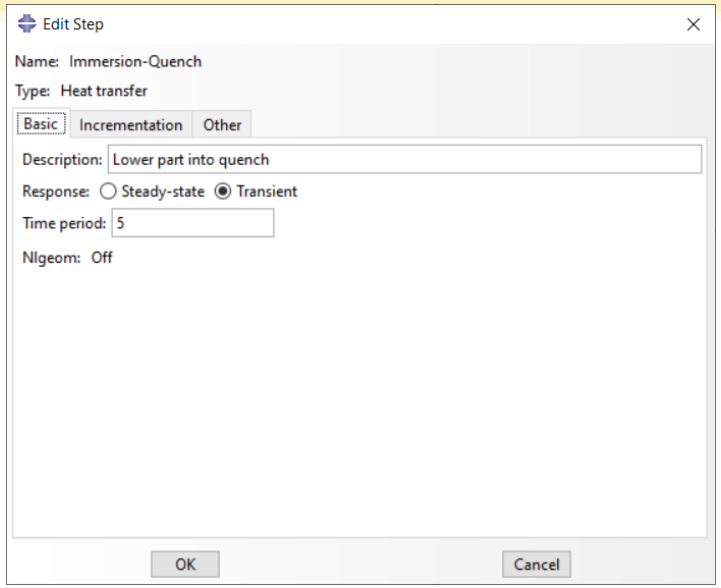


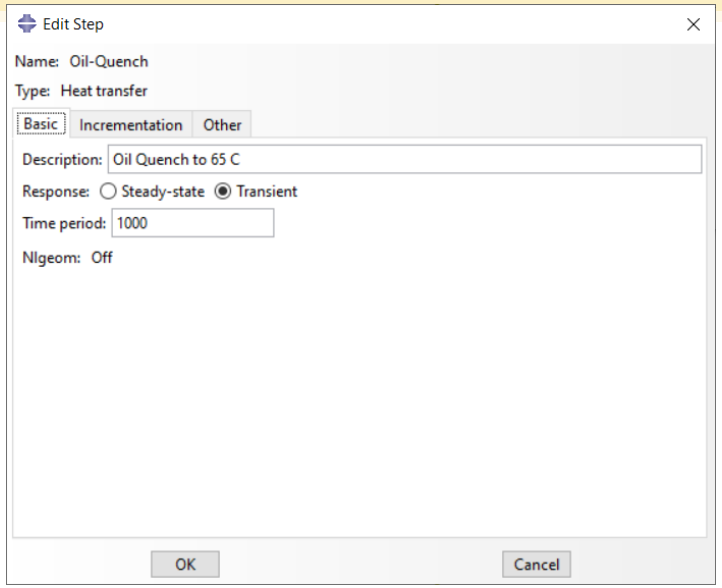


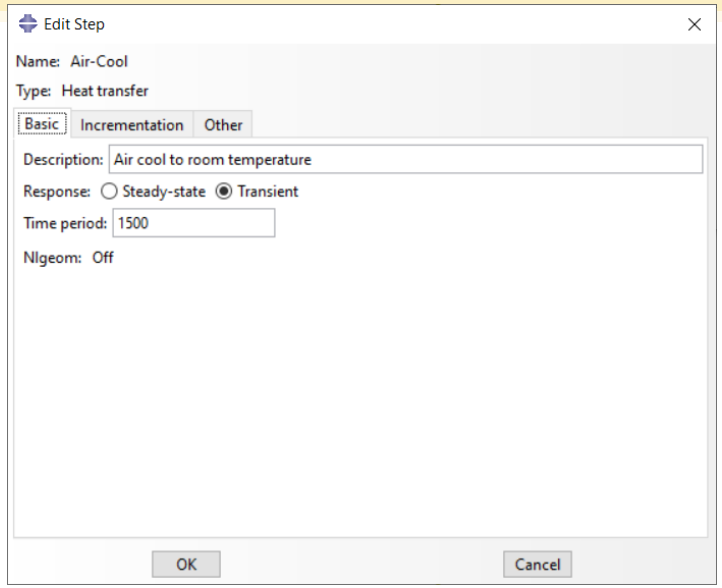


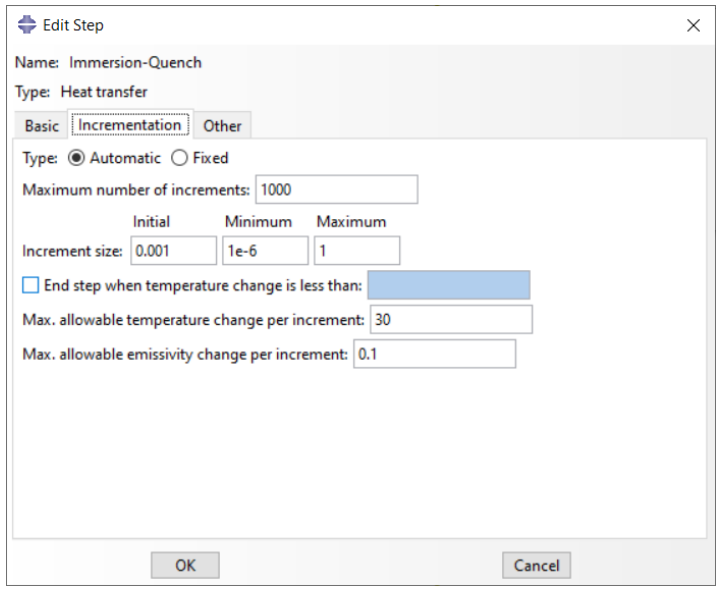


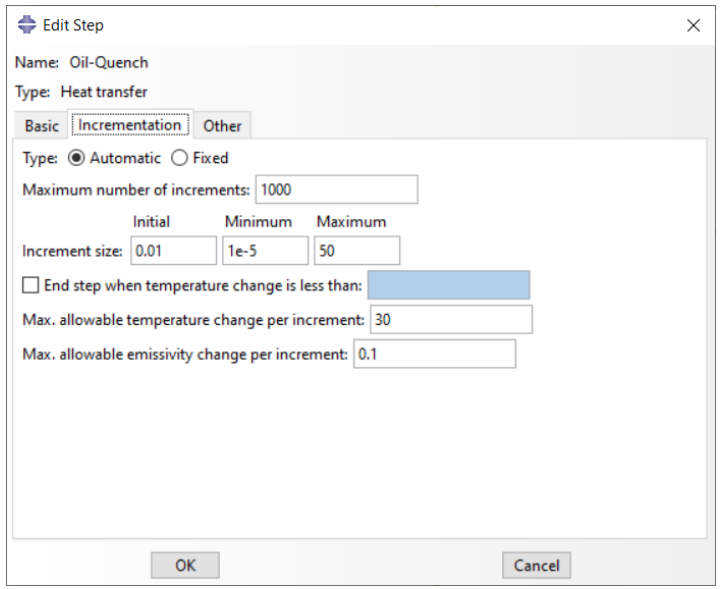


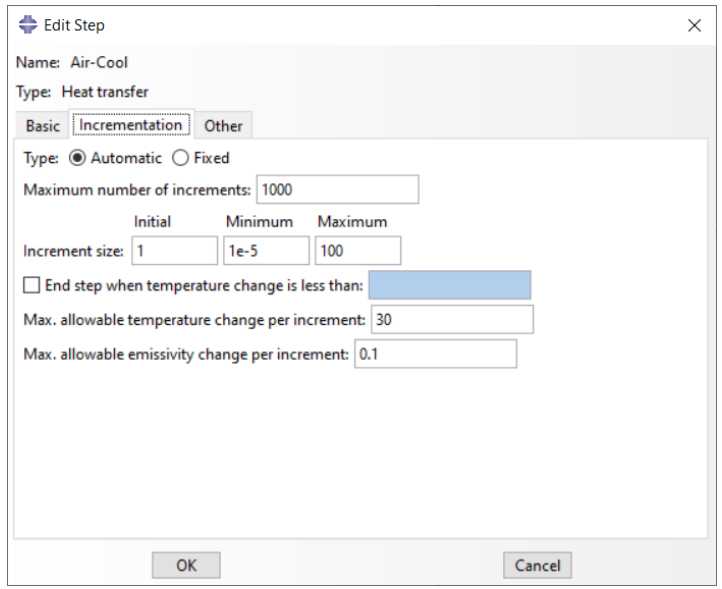




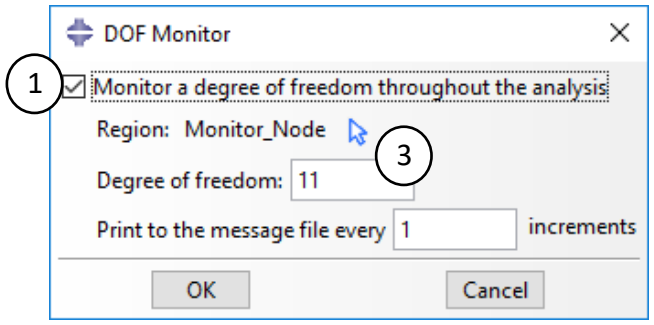


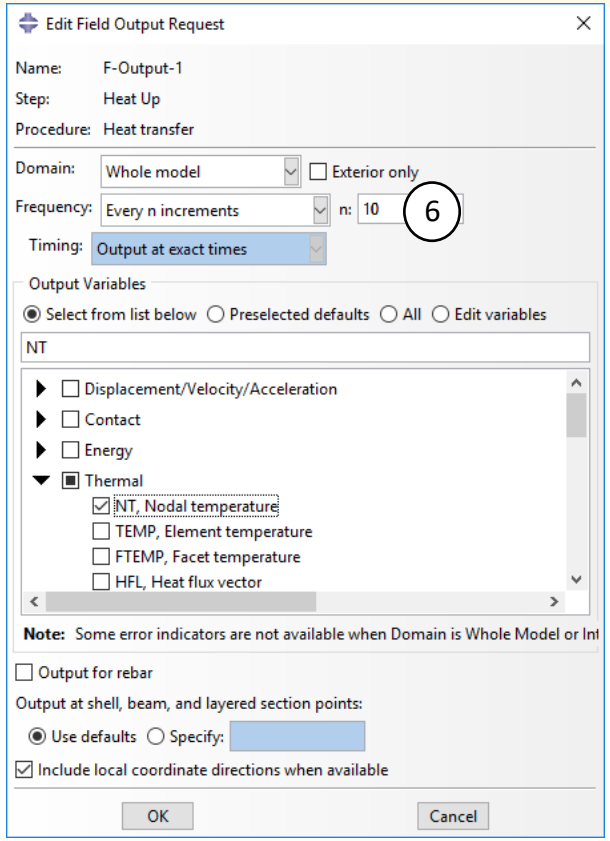


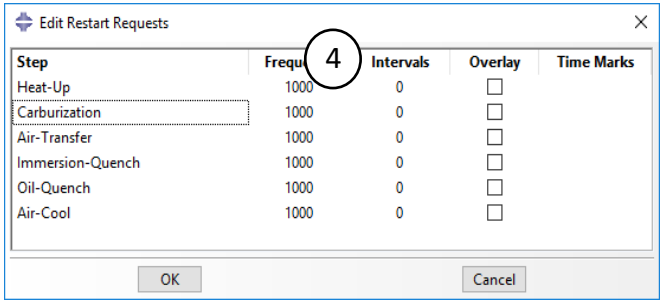




Step 3: Changing Output Definition

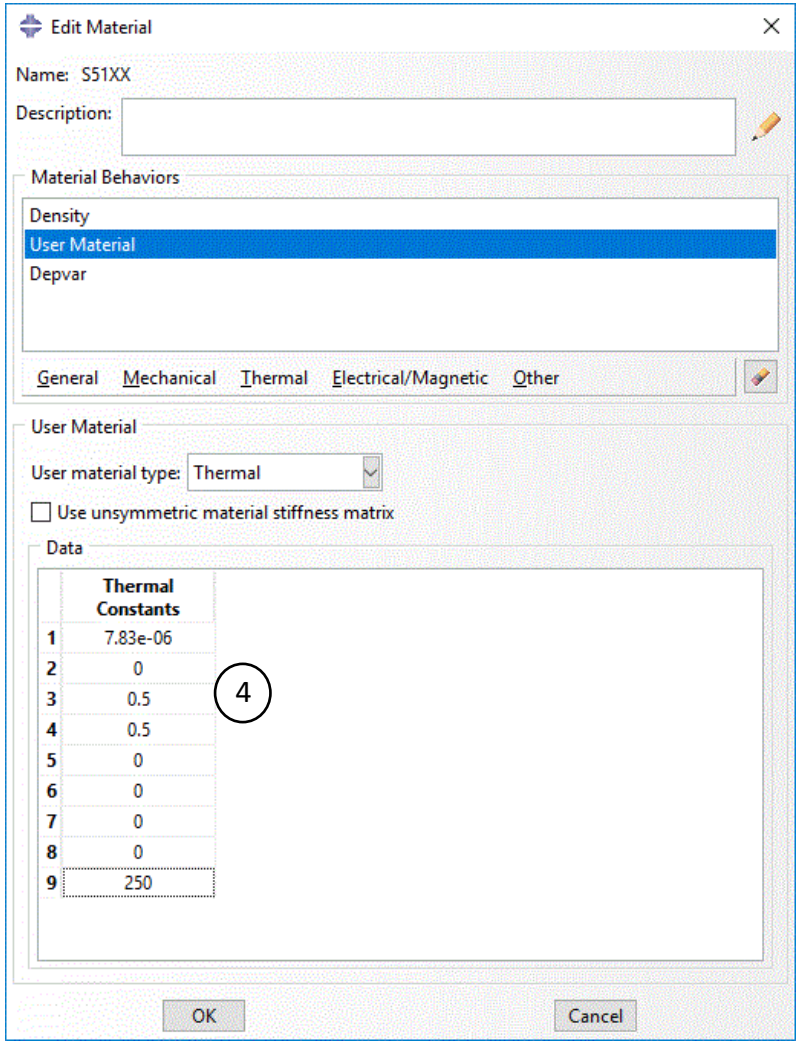




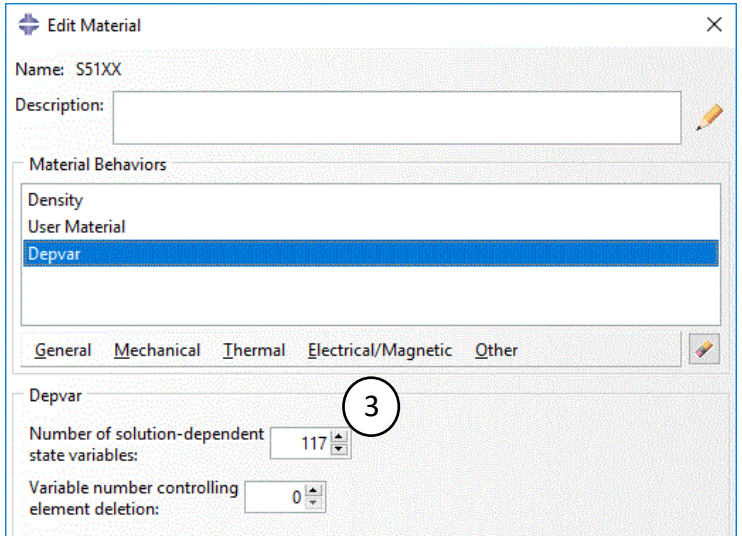


1. In the pulldown menu, select ***Output → DOF Monitor*** and check ***Monitor a degree of freedom throughout the analysis***
2. Select ***Monitor_Node*** which was created during the carburization model
3. Change the ***Degree of freedom*** to ***11*** and click ***OK***
4. In the pulldown menu, select ***Output → Restart Requests*** and change the ***Frequency*** for each step to ***1000***, then click ***OK***
5. Next, in the pulldown menu, select ***Output → Field Output Requests → Manager*** and click ***Edit*** for the ***Heat-Up*** step
6. Change the ***Frequency "n"*** to ***10*** and select the ***NT*** option

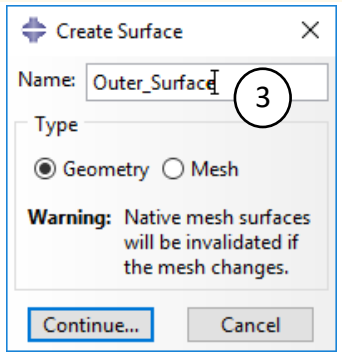
Step 4: Defining Thermal Material Properties

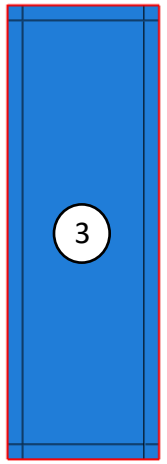


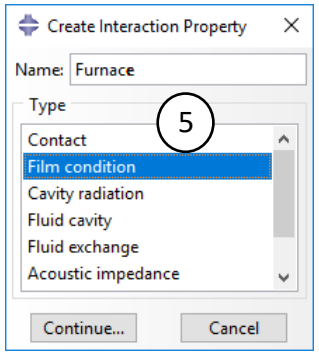
1. Under ***Module***, select ***Property***
2. In the pulldown menu, select ***Material*** → ***Manager*** and click ***Edit*** on the corresponding material (***S51XX***)
3. select ***General*** → ***Depvar*** and replace ***6*** with ***133*** under ***Number of solution-dependent state variables:***
4. Select ***User Material*** and enter ***7.83E-06***, ***0.0***, ***0.5***, ***0.5***, ***0.0***, ***0.0***, ***0.0***, ***0.0***, ***250, -1, 0.0635, -1, 0.8, 0.22, 0.15, 0.8, 0.04, -1, -1, -1, -1, -1, -1, -1, -1, -1, -1*** into cells 1-27. Cell 1 is the density, cell 2 is a flag to indicate a thermal model, cells 3 – 8 represent the initial microstructure, cell 9 represents the initial tempered martensite tempering temperature (if applicable), and cells 10 – 27 represent the alloy composition of the material. Please see the help file for more information on these values. Then click ***OK***.

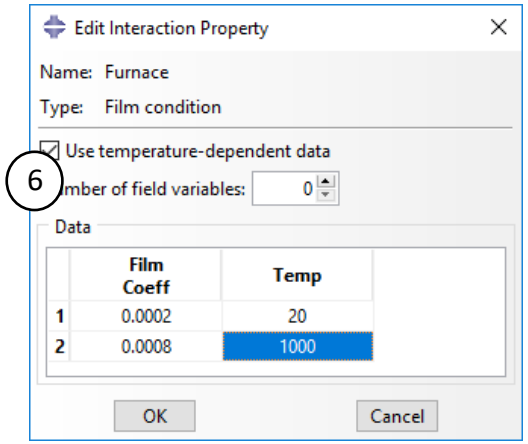


Step 5: Applying Thermal Boundary Conditions





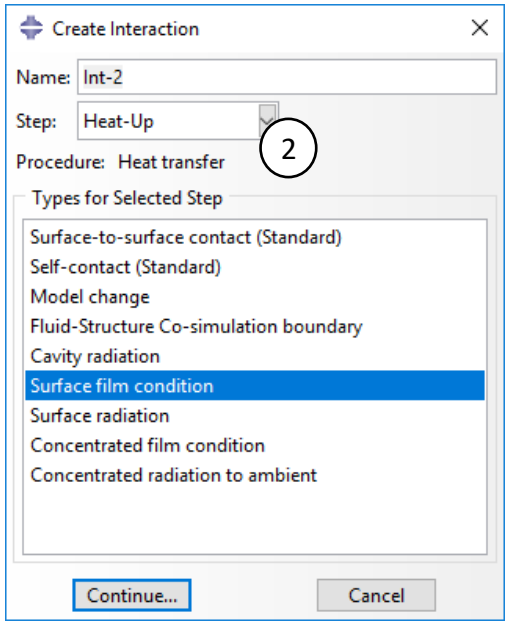


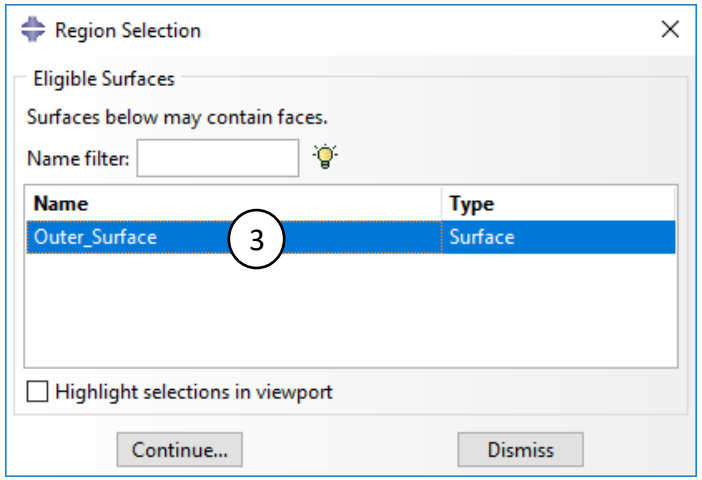


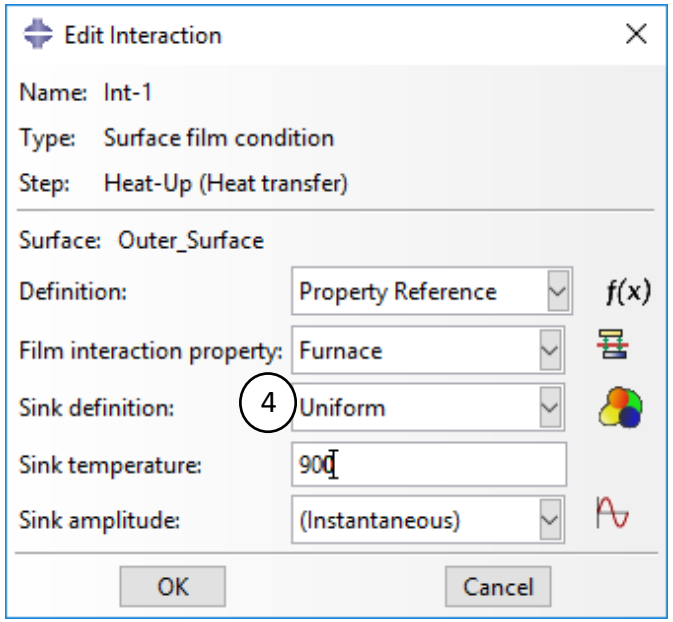

```
o air-cool
   0.00011,  37.
   0.000129, 65.
   0.000143, 93.
   0.000162, 120.
   0.000178, 148.
   0.000197, 176.
   0.000237, 231.
   0.000283, 287.
   0.000335, 343.
   0.000393, 398.
   0.000458, 454.
   0.000533, 509.
   0.000616, 565.
   0.000708, 620.
   0.000811, 676.

o Air-transfer                    6
   0.000237,    231.
   0.000283,    287.
   0.000335,    343.
   0.00010,     400.
   0.00010,     500.
   0.00009,     565.
   0.00008,     620.
   0.00008,     850.0

o furnace
   0.0002, 20.
   0.0008, 1000.

o oil-quench
   0.0001, 20.
   0.0005, 150.
   0.0015, 300.
   0.002, 400.
   0.00375, 450.
   0.005, 500.
   0.005, 550.
   0.00475, 600.
   0.003, 650.
   0.002, 700.
   0.0015, 750.
   0.0013, 800.
```

1. Under ***Module***, select ***Interaction***
2. Define the outer surface by selecting ***Tools*** → ***Surface*** → ***Create*** in the pulldown menu
3. Name the surface ***Outer_Surface,*** check ***Geometry***, click ***Continue***, box select the entire region, and then click ***Done***
4. Go to ***Interaction*** → ***Property*** → ***Manager*** and delete any previously created ***Interactions***
5. Click ***Create***, name the first ***Interaction Property***, ***Furnace***, select ***Film Condition***, and click ***Continue***
6. Check ***Use temperature-dependent data*** and input the provided values for the ***Furnace*** step. The ***Air-Cool***, ***Air-Transfer***, ***Furnace***, and ***Oil-Quench*** film condition follows the same steps with the data provided to the left. Each row contains one data pair, with the first number being the ***Film Coefficient*** and the second number being the ***Temperature***.

Step 5b: Applying Thermal Boundary Conditions

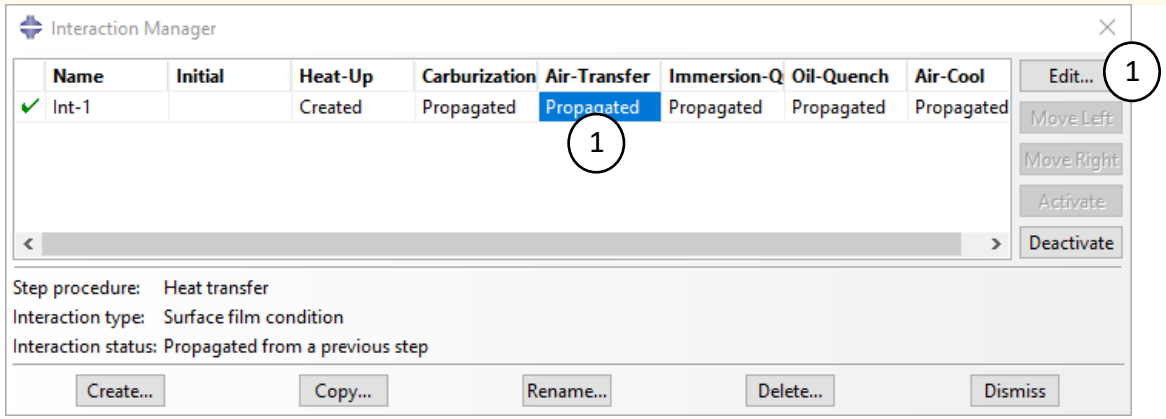




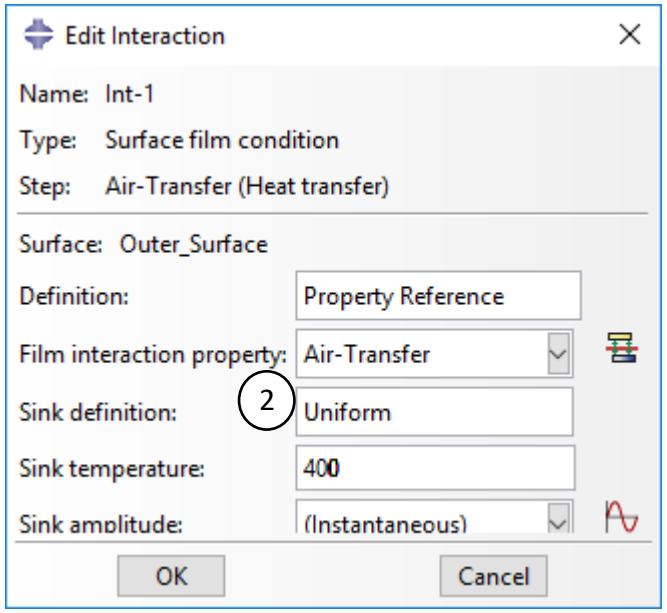


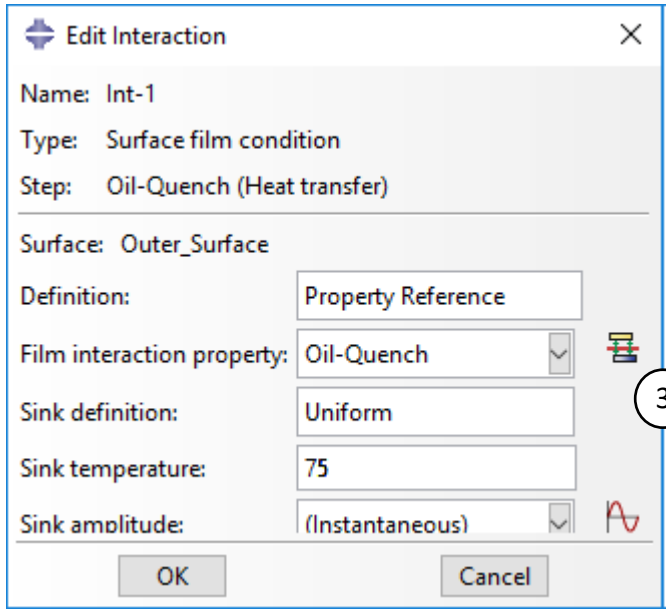
1. In the pulldown menu, select ***Interaction*** → ***Manager*** and delete any previous ***Interactions*** if applicable
2. Click ***Create***, select the first step ***Heat-Up***, select ***Surface film condition***, then click ***Continue***
3. Select the predefined surface called ***Outer_Surface*** and click ***Continue***
4. Select ***Property Reference*** in the ***Definition*** tab, select the corresponding ***Film Condition*** (***Furnace***), write in ***900*** for ***Sink temperature***, then click ***OK***

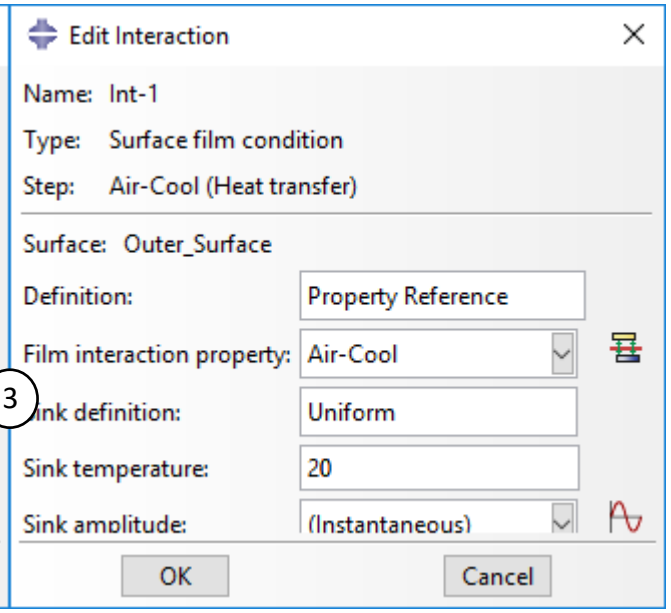
Step 5c: Applying Thermal Boundary Conditions



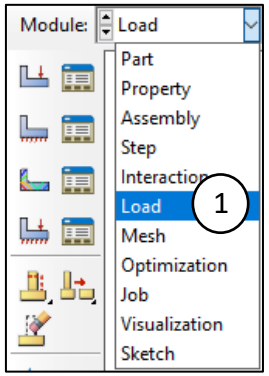
1. In the ***Interaction Manager*** window, highlight the next step with a different thermal BC and click ***Edit*** (***Air Transfer***)
2. Change the ***Film interaction property*** as well as the ***Sink temperature*** to the correct conditions
3. Repeat these steps for the ***Oil-Quench and Air-Cool*** steps.
   - ***Immersion-Quench*** will not be edited

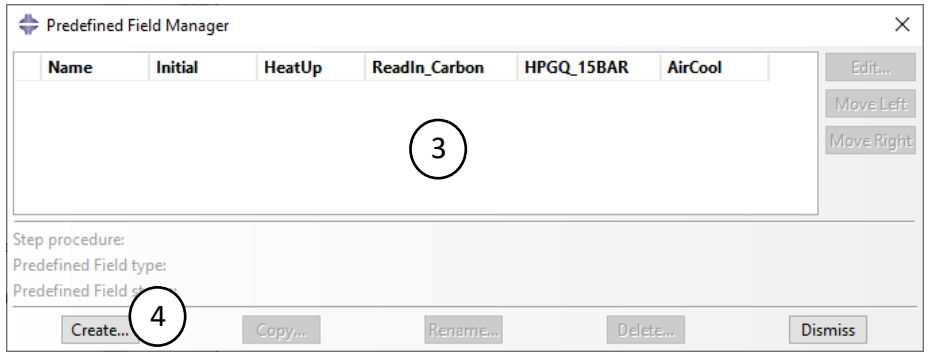




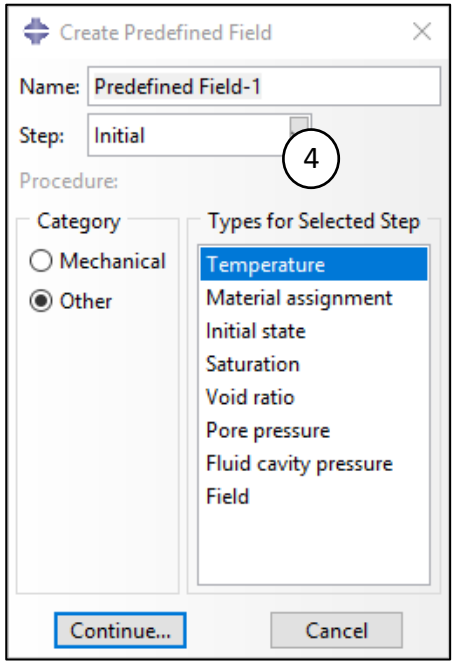


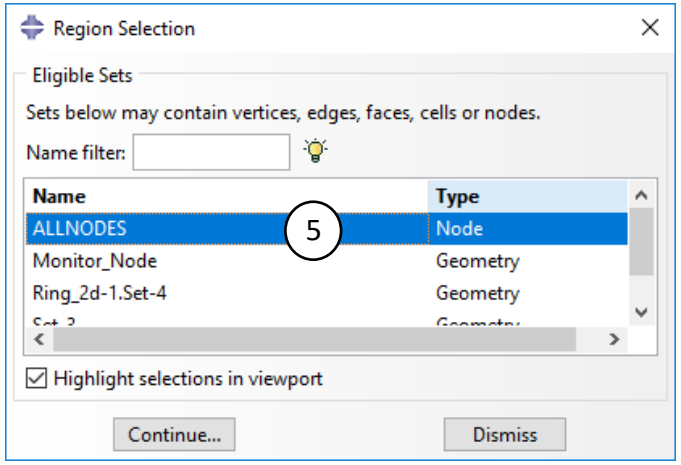
Step 6: Changing Initial Temperature Condition

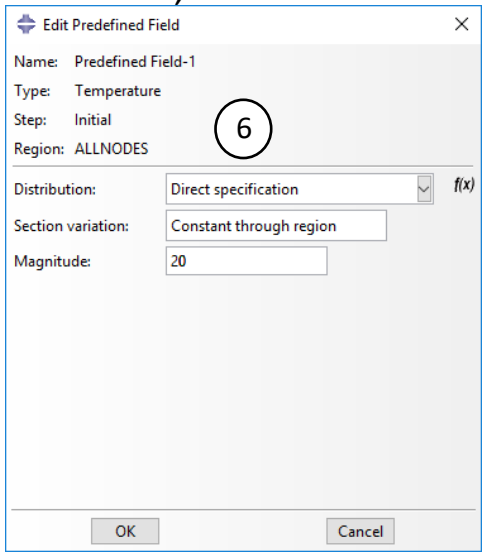


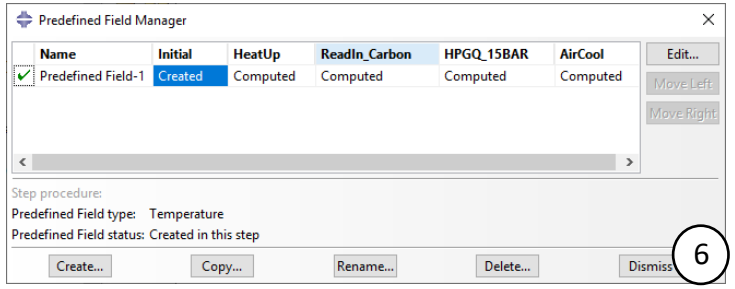


1. Under ***Module***, select ***Load***
2. Create an ***ALLNODES*** set if not currently present ***(Tools → Set → Create)***
3. In the pulldown menu, go to ***Predefined Field → Manager*** and delete any ***Predefined Fields*** that remain.
4. Click ***Create***, select ***Initial*** under the ***Step:*** tab, check ***Other***, select ***Temperature*** and click ***Continue***
5. Select ***ALLNODES*** and click ***Continue***
6. Type ***20*** in ***Magnitude*** to represent initial temperature then click ***OK***, then ***Dismiss***

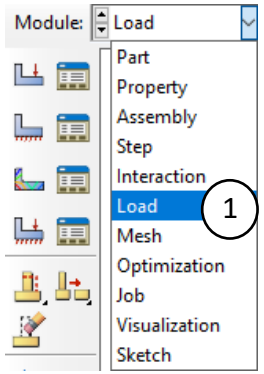


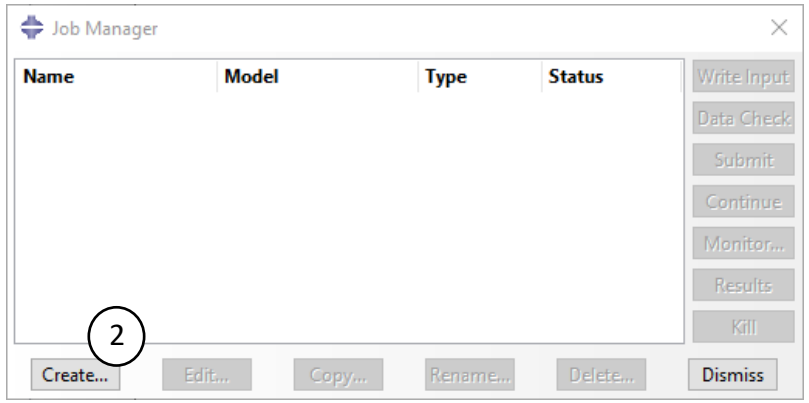




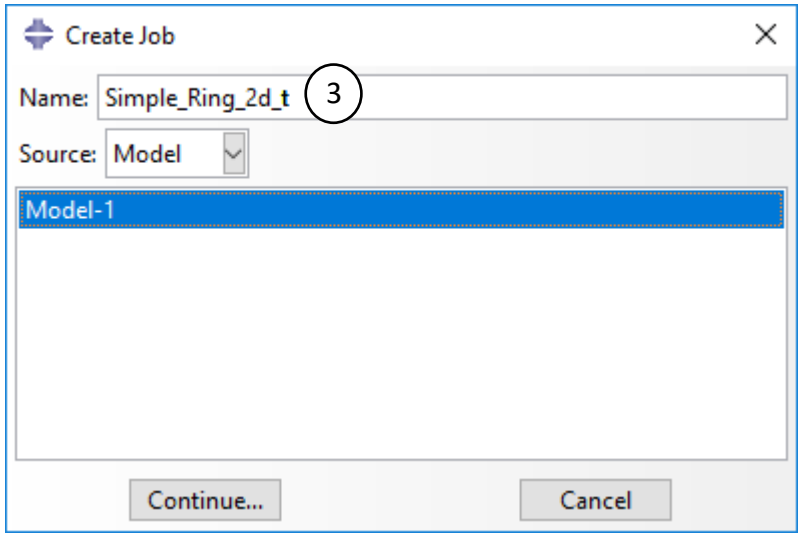


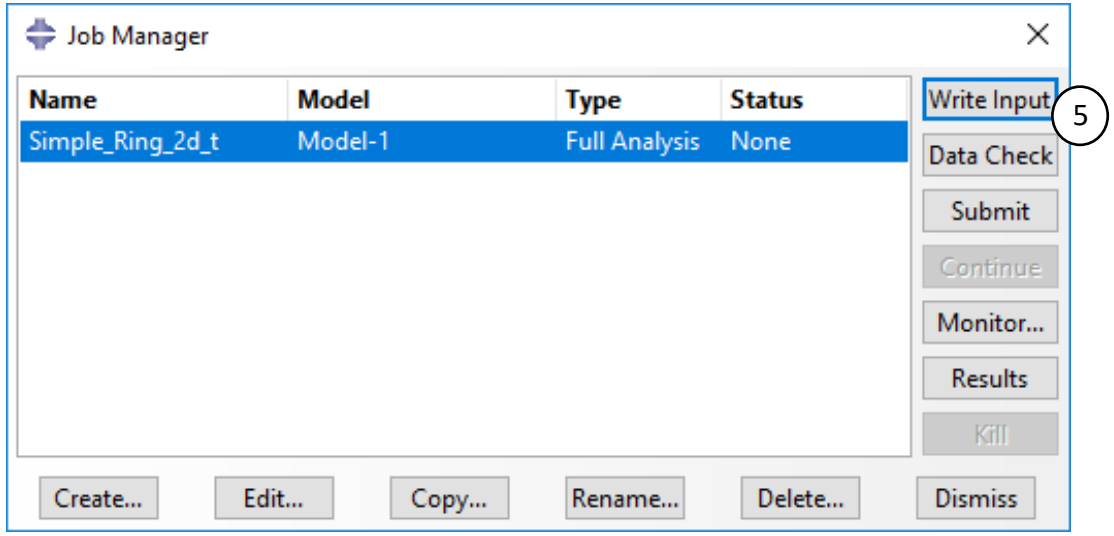
Step 7: Writing Out the Input File





1. Under ***Module***, go to ***Job***
2. In the pulldown menu, select ***Job*** → ***Manager*** and click ***Create***
3. Name the job ***Simple_Ring_2d_t ,*** then click ***Continue***
4. Use the default values and click ***OK***
5. Click on ***Write Input*** to write out the .inp file to be edited





Step 8: Editing The Input File

```
**
*Material, name=S51XX
*Density
 7.83e-06,
*Depvar
    133,
    1,   DEFWF_CARB,          Defined Nascent Carbon Weight Fraction
    2,   HARDNESS,            Total Hardness
    4,   DEFWF_NITROGEN,      Defined Nascent Nitrogen Weight Fraction
    21,  VF_AUSTENITE,        Volume Fraction of Austenite
    34,  VF_FERRITE,          Volume Fraction of Ferrite
    47,  VF_PEARLITE,         Volume Fraction of Pearlite
    60,  VF_UBAINITE,         Volume Fraction of Upper Bainite
    73,  VF_LBAINITE,         Volume Fraction of Lower Bainite
    86,  VF_MARTENSITE,       Volume Fraction of Martensite
    99,  VF_TMARTENSITE,      Volume Fraction of Tempered Martensite
    102, WF_CARB_AUST,        Carbon Weight Fraction in Austenite
    103, WF_CARB_FERR,        Carbon Weight Fraction in Ferrite
    104, WF_CARB_PEARL,       Carbon Weight Fraction in Pearlite
    105, WF_CARB_UBAIN,       Carbon Weight Fraction in Upper Bainite
    106, WF_CARB_LBAIN,       Carbon Weight Fraction in Lower Bainite
    107, WF_CARB_MART,        Carbon Weight Fraction in Martensite
    108, WF_CARB_TMART,       Carbon Weight Fraction in Tempered Martensite
    109, DEFWF_CBDA,          Defined Carbon Weight Fraction in Carbide A
    110, DEF_CBDA_SIZE,       Defined Carbide A Size Factor
    113, DEFWF_NTDA,          Defined Nitrogen Weight Fraction in Nitride A
    114, DEF_NTDA_SIZE,       Defined Nitride A Size Factor
*User Material, constants=27, type=THERMAL
 7.83e-06,     0.,    0.3,    0.7,     0.,     0.,     0.,     0.
   250.,    -1., 0.0635,    -1.,    0.8,   0.22,   0.15,    0.8
   0.04,    -1.,    -1.,    -1.,    -1.,    -1.,    -1.,    -1.
    -1.,    -1.,    -1.
```

1

1. Once the input file has been opened, go to the ***MATERIALS*** section and define the ***Solution-Dependent State Variable (SDV)*** to be analyzed as shown below. Simply copy and paste the text below into the input file.

```
1,   DEFWF_CARB,          Defined Nascent Carbon Weight Fraction
2,   HARDNESS,            Total Hardness
4,   DEFWF_NITROGEN,      Defined Nascent Nitrogen Weight Fraction
21,  VF_AUSTENITE,        Volume Fraction of Austenite
34,  VF_FERRITE,          Volume Fraction of Ferrite
47,  VF_PEARLITE,         Volume Fraction of Pearlite
60,  VF_UBAINITE,         Volume Fraction of Upper Bainite
73,  VF_LBAINITE,         Volume Fraction of Lower Bainite
86,  VF_MARTENSITE,       Volume Fraction of Martensite
99,  VF_TMARTENSITE,      Volume Fraction of Tempered Martensite
102, WF_CARB_AUST,        Carbon Weight Fraction in Austenite
103, WF_CARB_FERR,        Carbon Weight Fraction in Ferrite
104, WF_CARB_PEARL,       Carbon Weight Fraction in Pearlite
105, WF_CARB_UBAIN,       Carbon Weight Fraction in Upper Bainite
106, WF_CARB_LBAIN,       Carbon Weight Fraction in Lower Bainite
107, WF_CARB_MART,        Carbon Weight Fraction in Martensite
108, WF_CARB_TMART,       Carbon Weight Fraction in Tempered Martensite
109, DEFWF_CBDA,          Defined Carbon Weight Fraction in Carbide A
110, DEF_CBDA_SIZE,       Defined Carbide A Size Factor
113, DEFWF_NTDA,          Defined Nitrogen Weight Fraction in Nitride A
114, DEF_NTDA_SIZE,       Defined Nitride A Size Factor
```

Step 8b: Editing The Input File

1. Go to the ***PREDEFINED FIELDS*** section of the file and add the initial conditions as seen in the image to the right
2. Next, in the ***Carburization*** step, add the two lines as seen in the bottom right to simulate the addition of carbon
    - The file name given in the ***INPUT*** must be the same as the .nod file name in the working directory created during the post processing of the carburization model

```
** PREDEFINED FIELDS
**
** Name: Predefined Field-1   Type: Temperature
*Initial Conditions, type=TEMPERATURE
ALLNODES, 20.
*Initial Conditions, type=Field, VAR=1
ALLNODES, -4
*Initial Conditions, type=Field, VAR=3
ALLNODES, 0.002
*Initial Conditions, type=Field, VAR=12
ALLNODES, 0.0
** ----------------------------------------------------------------
**
** STEP: HeatUp
**
*Step, name=HeatUp, nlgeom=NO, inc=1000, amp=step
*Heat Transfer, end=PERIOD, deltmx=30.
0.01, 1800., 1e-06, 1800.,
```

1

```
4908 ** ----------------------------------------------------------------
4909 **
4910 ** STEP: Carburization
4911 **
4912 *Step, name=Carburization, nlgeom=NO, inc=1000,
4913 *Heat Transfer, end=PERIOD, deltmx=30.
4914 1., 10620., 5e-06, 500.,
4915 **
4916 *FIELD, OP=NEW, VAR=3, INPUT=simple_ring_cc.nod
4917 *Sfilm,
4918 ** OUTPUT REQUESTS
```

2

Step 9: Editing the Input File Cont.

```
4960 ** STEP: Immersion-Quench
4961 **
4962 *Step, name=Immersion-Quench, nlgeom=NO, inc=1000, amp=step
4963 *Heat Transfer, end=PERIOD, deltmx=30.
4964 0.001, 5., 1e-06, 1.,
4965 **
4966 *Sfilm,
4967 Outer_Surface, FNU
4968 ** OUTPUT REQUESTS
4969 **
```

(2) (1)

```
4882 ** STEP: Heat-Up
4883 **
4884 *Step, name=Heat-Up, nlgeom=NO, inc=1000, amp=step
4885 *Heat Transfer, end=PERIOD, deltmx=30.
4886 1., 1000., 1e-06, 100.,
4887 **
```

(3)

```
4910 ** STEP: Carburization
4911 **
4912 *Step, name=Carburization, nlgeom=NO, inc=1000, amp=ramp
4913 *Heat Transfer, end=PERIOD, deltmx=30.
4914 1., 10620., 5e-06, 500.,
```

(3)

```
4888 ** INTERACTIONS
4889 **
4890 ** Interaction: Int-1
4891 *Sfilm, op=new
4892 Outer_Surface, F, 900., Furnace
4893 **
```

(4)

1. In the ***Immersion-Quench*** step, add the two lines as seen to the left
2. Add an ****inc=1000*** to the any ****Step*** without an increment defined
3. For each ****Step*** definition, add ***amp=step*** at the end except for ***Carburization*** where ***amp=ramp*** will be added
4. For each ****Sfilm*** definition, add ***op=new*** at the end of the line
5. Next, in each step, add the 3 lines shown below to output the ***Solution-Dependent State Variables (SDV)*** defined earlier

****Element Output, directions=YES***

SDV1, SDV2, SDV4, SDV21, SDV34, SDV47, SDV60, SDV73, SDV86, SDV99, SDV102, SDV103

SDV104, SDV105, SDV106, SDV107, SDV108, SDV109, SDV110, SDV113, SDV114

- Make sure to write out this line directly after ****Node Output*** line

```
*Restart, write, frequency=1000
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, frequency=10
*Node Output
NT,
*Element Output, directions=YES
SDV1, SDV2, SDV4, SDV21, SDV34, SDV47, SDV60, SDV73, SDV86, SDV99, SDV102, SDV103
SDV104, SDV105, SDV106, SDV107, SDV108, SDV109, SDV110, SDV113, SDV114
**
*Output, history, frequency=0
*Node File, FREQ=1
  NT
*End Step
```

(4)

Step 9b: Editing The Input File

```
4880 ** ----------------------------------------------------------------
4881 **
4882 ** STEP: Heat-Up
4883 **
4884 *Step, name=Heat-Up, nlgeom=NO, inc=1000, amp=step
4885 *Heat Transfer, end=PERIOD, deltmx=30.
4886 1., 1000., 1e-06, 100.,
4887 **
4888 *CONTROLS, PARAMETERS=LINE SEARCH
4889 6,
4890 *CONTROLS, PARAMETERS=TIME INCREMENTATION
4891 20, 30
4892 *CONTROLS, FIELD=TEMPERATURE, PARAMETERS=FIELD
4893 0.05,0.05,
4894 **
4895 ** INTERACTIONS
4896 ** Interaction: Int-1
4897 *Sfilm, op=new
4898 Outer_Surface, F, 900., Furnace
4899 **
4900 ** OUTPUT REQUESTS
4901 **
```

(1)

1. Add the lines as seen in the image to the left to the first step
   - These lines will help with convergence
2. Also add the lines seen in the image to bottom right to every step to output the ***Nodal Temperatures*** which will be used in the stress model

```
4900 ** OUTPUT REQUESTS
4901 **
4902 *Restart, write, frequency=1000
4903 *Monitor, dof=11, node=Monitor_Node, frequency=1
4904 **
4905 ** FIELD OUTPUT: F-Output-1
4906 **
4907 *Output, field, frequency=10
4908 *Node Output
4909 NT,
4910 *Element Output
4911 SDV1, SDV2, SDV4, SDV5, SDV6, SDV7, SDV8, SDV9, SDV21, SDV34, SDV47, SDV60, SDV73, SDV86, SDV99, SDV117
4912 *Output, history, frequency=0
4913 *NODE FILE, FREQ=1
4914  NT
4915 *End Step
4916 ** ----------------------------------------------------------------
4917 **
4918 ** STEP: Carburization
```

(2)

Step 9c: Editing The Input File

```
**=================================================================
** File Name: "Model Name" + "_FILM-QUENCH.TXT"
**=================================================================
** 1: Linear Quench (Immersion or Spray)
** 2: Rotational Quench (Spray)
** 3: Gas Quench
**
*ITYPE_QUENCH
 1
** The following format is for Linear Quench
** Number of Surfaces to define the HTC and ambient (<=10 allowed)
*NUM_SURF_TOT
 1
**--Surface Name*****************************************************
*START_SURFACE
**
*SURFACE_NAME
 Outer_Surface
**
**--Direction of immersion (0,0,0)-->(x1,y1,z1)---------------
*QUENCH_DIRECTION
 0.0, -1.0, 0.0
** Speed <-->Time (mm/s)
*TRAVEL_SPEED
 12
** Quenching starts right away or starts at certain time
*QSTART_TIME
 0.0
** Can be less than 0
*QSTART_LINE
 0.0
** Negative number means infinity
*QEND_LINE
 -10.0
*QEFFECT_LINE
 0.0
**
**--Ambient temperature of Air--------------------------------
*TAMBT_AIR
20.0
**--Heat transfer coefficient of Air [W/(mm^2C)]--------------
*HTC_AIR
0.000500
**
**--Temperature of Effect Zone--------------------------------
*TAMBT_EFFECT
20.0
**--HTC of Effect Zone [W/(mm^2C)]--------------
*HTC_EFFECT
1.0E-03
**
**--Temperature of quenchant [Unit: Degree C]---------
*TAMBT_QUENCH
 75
**--HTC of quenchant--------------------
*HTC_QUENCH
 0.00010,   20.0
 0.00050,  150.0
 0.00150,  300.0
 0.00200,  350.0
 0.00250,  400.0
 0.00375,  450.0
 0.00500,  500.0
 0.00500,  550.0
 0.00475,  600.0
 0.00300,  650.0
 0.00200,  700.0
 0.00150,  750.0
 0.00130,  800.0
 0.00050,  850.0
 0.00080, 1000.0
*END_SURFACE
**End of File
```

4

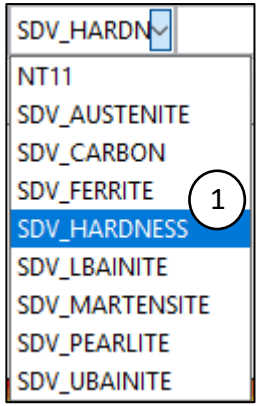
1. The ***Immersion-Quench Text File*** must be prepared in order to simulate Immersion Quenching
2. This file is included in the directory with this tutorial
3. The file name must have the format

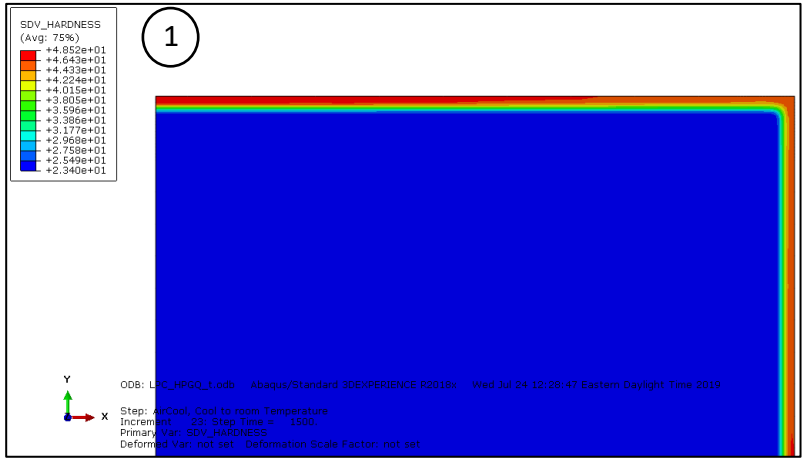
   “ModelName “ + “_ “+ “FILM-QUENCH.TXT”

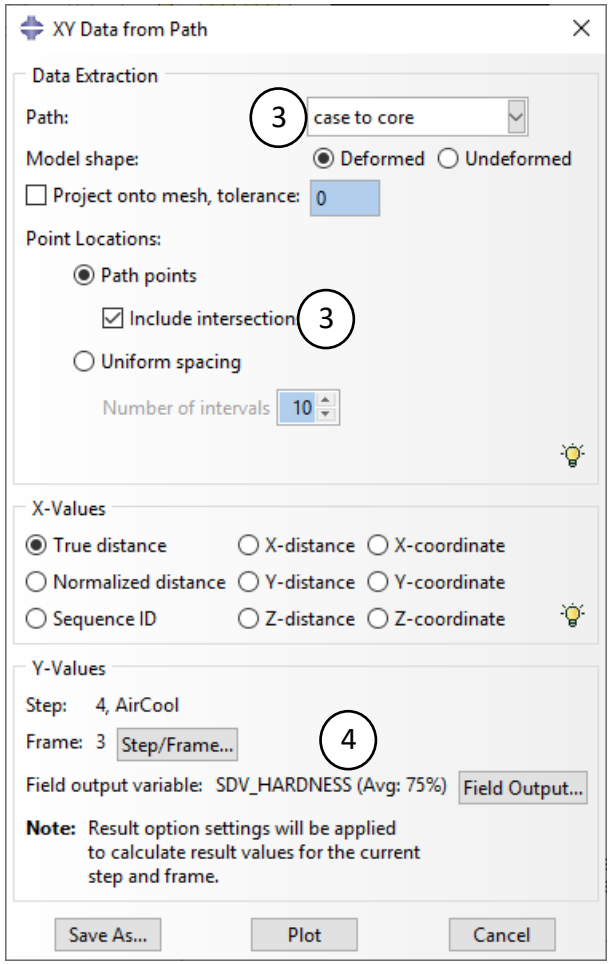
   following this tutorial’s naming, Simple_Ring_2d_t_FILM-QUENCH.TXT

3. The file must be in the same working directory as the input file
4. Be sure to check the ***Immersion-Quench Text File*** so that the values and definitions in both the ***Input File*** and the ***Immersion-Quench File*** match. (such as the outer surface name)
6. Save the input file and submit the job through the command line using the same method as the carburization model.
7. To open the file, go back to ***Abaqus Cae***, under ***Module***, select ***Visualization*** and open the .odb file located in the working directory
8. As with all simulations, take a moment and check the results to ensure that they make sense. We expect temperatures to reach 900°C at the end of Heat-Up, and ~20°C at the end of air cool

Step 10: Verifying the Results/Path Plots

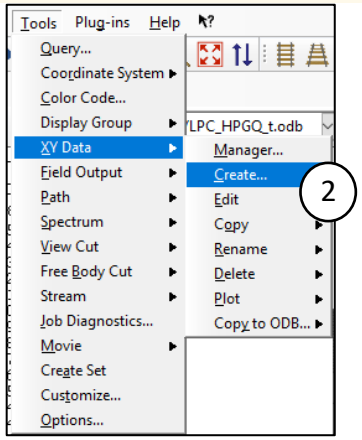


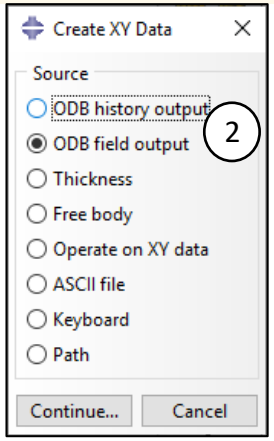


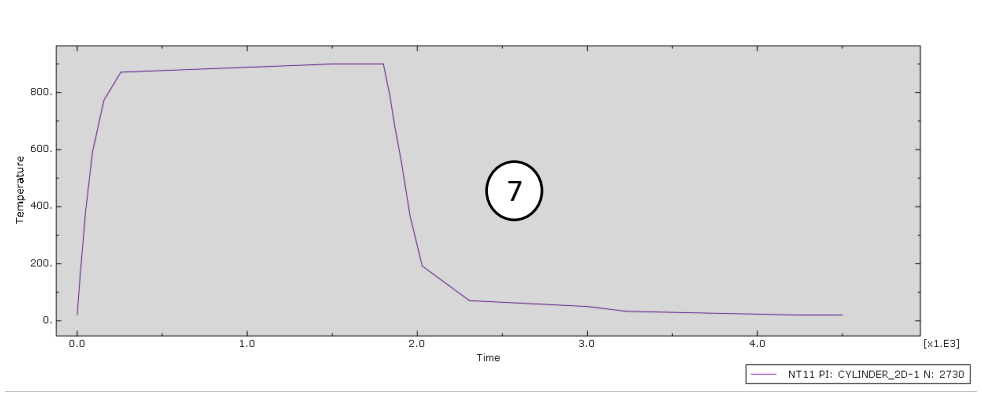


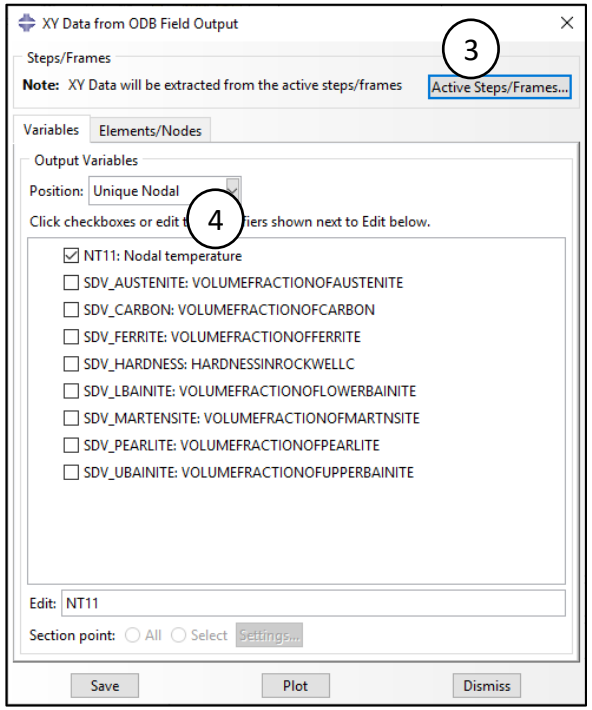
1. In the Field Output menu, one can select several DANTE solution dependent variables to display on the contour. While the Abaqus NT11 variable represents temperature in the thermal model, one could also display hardness, carbon, austenite, ferrite, upper and lower bainite, pearlite, martensite, and if the kinetics are set for it(-10 or -11), tempered martensite
2. A path plot of the hardness or phases is often desirable to see harness vs. depth to ensure good wear/fracture protection. Set up the case to core path the same way as in the carburization post-processing
3. 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 Hardness vs. Depth plot
4. To plot a different variable, select the desired field in the *XY Data from Path* window. One can also plot the fields at a different time than the end of the process by selecting the desired step and frame in the same window

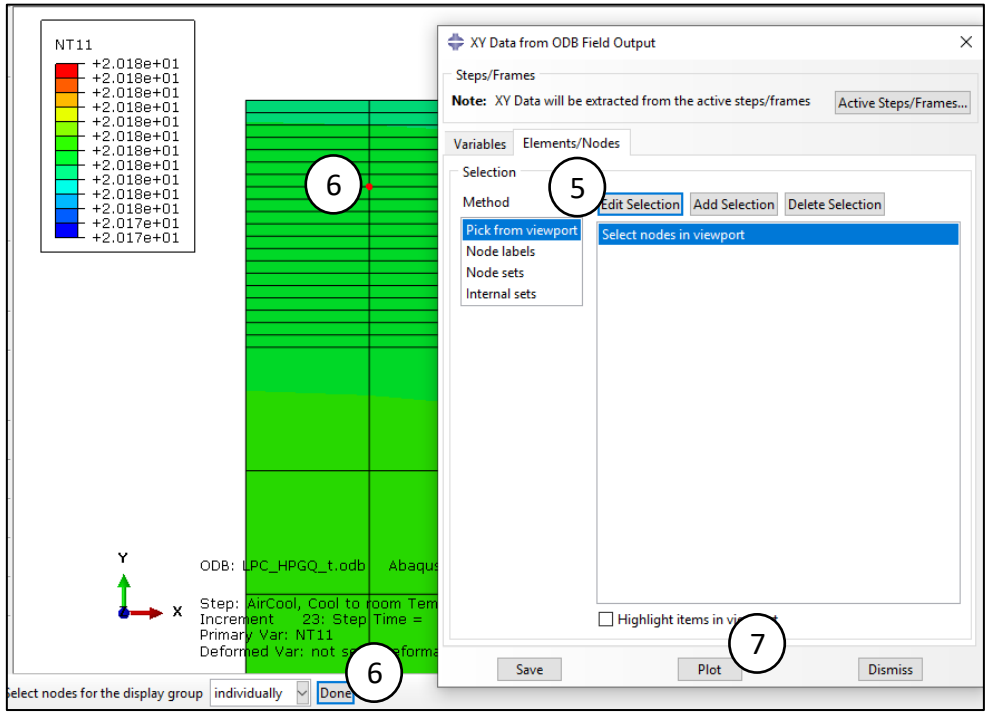
Step 11: Post Processing (History Plot)







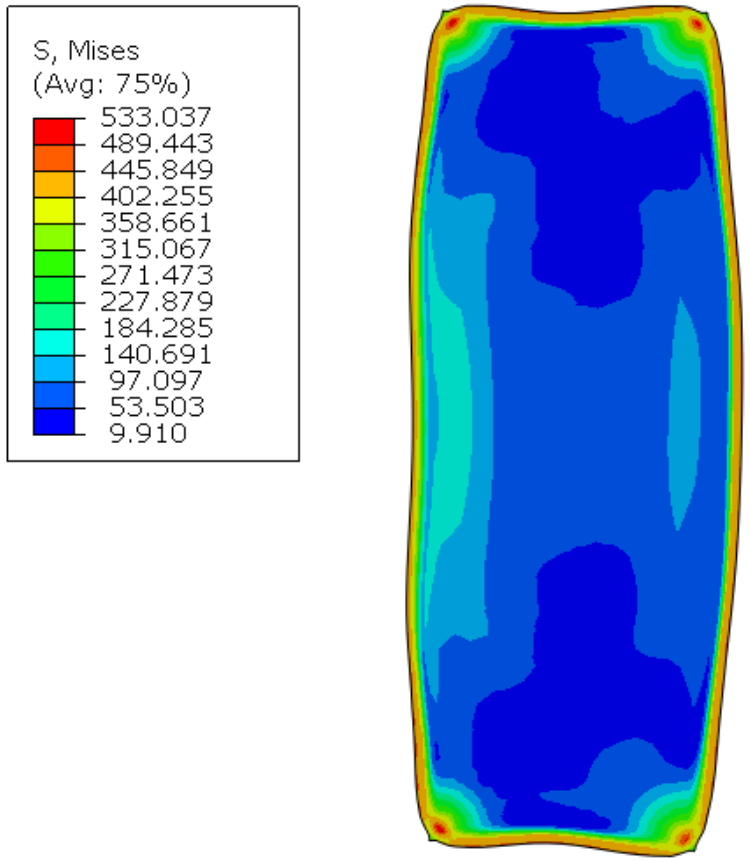




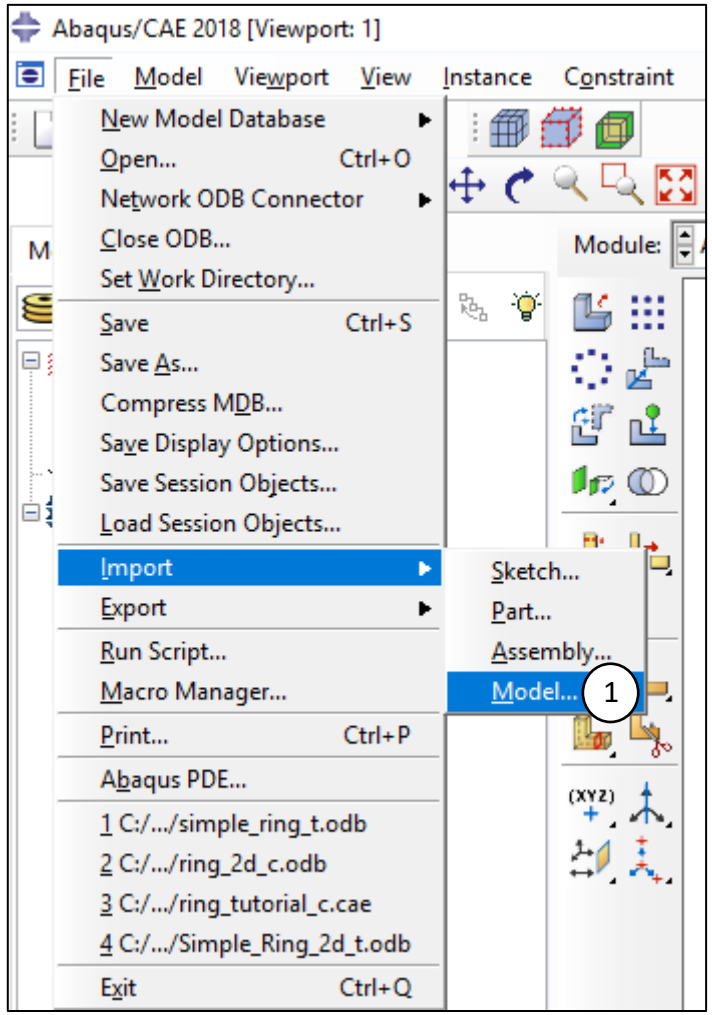
1. Temperature history is another useful tool for post processing a thermal file as one can follow a node through the entire thermal history of a process in order to get an understanding of temperatures and cooling rates on that node/section.
2. In the drop-down menu select ***Tools → XY Data → Create*** to open the *Create XY Data* window. Select ODB field output and click continue.
3. At the top of the following window one can change which active steps or frames they desire to be plotted; default is all.
4. For temperature, Select Unique Nodal in the output variables section, and check the box next to NT11: Nodal Temperature.
5. In the Elements/Nodes tab, check that you are on *pick from viewport* and click ***Edit Selection***.
6. Select the node to be plotted and click Done, or middle mouse click, in the viewport.
7. Click plot in the *XY Data from ODB Field Output window* to show the thermal history of that node throughout the process.

Note: Exporting this data to plot in excel follows the same method as saving the carbon .nod file in the previous section. (Report -> XY …)

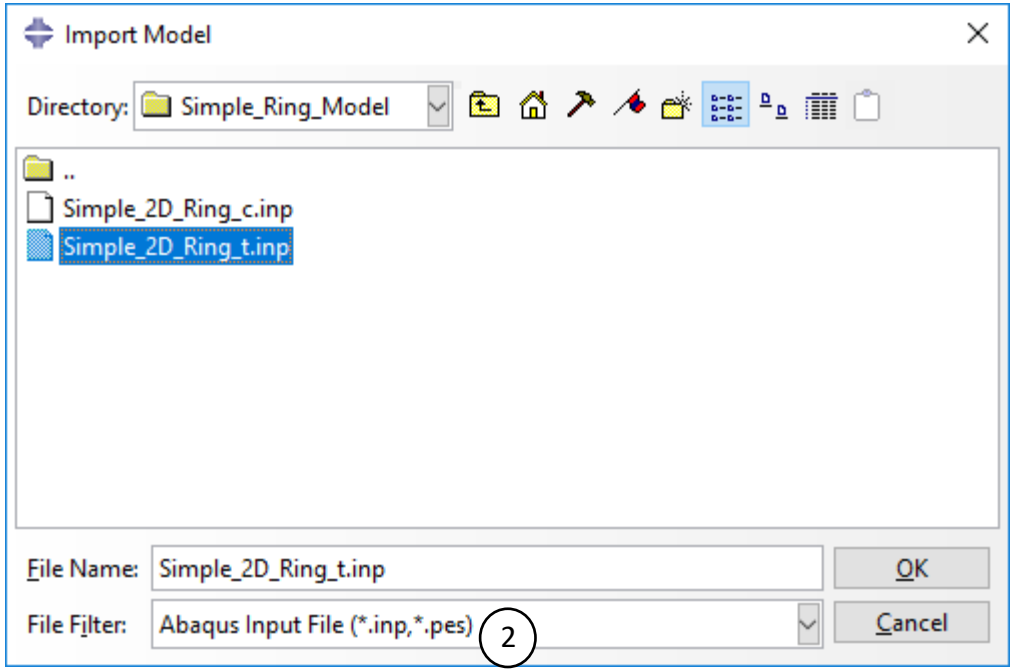
Stress Model

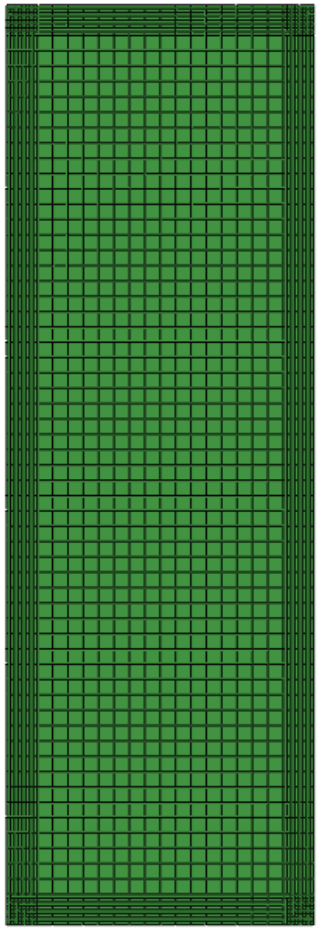


Step 1: Importing Thermal Model

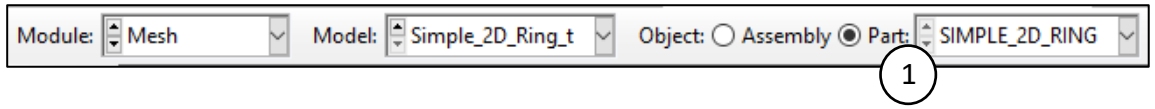


1. In the pulldown menu, select ***File → Import → Model***
2. Change the ***File Filter*** option to ***Abaqus Input File*** and import the thermal input file

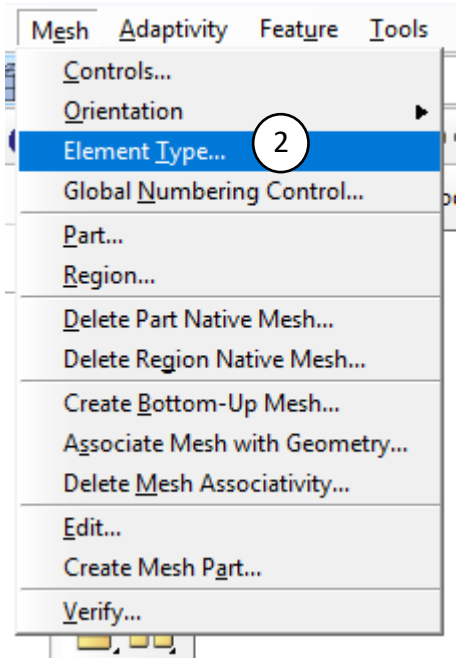


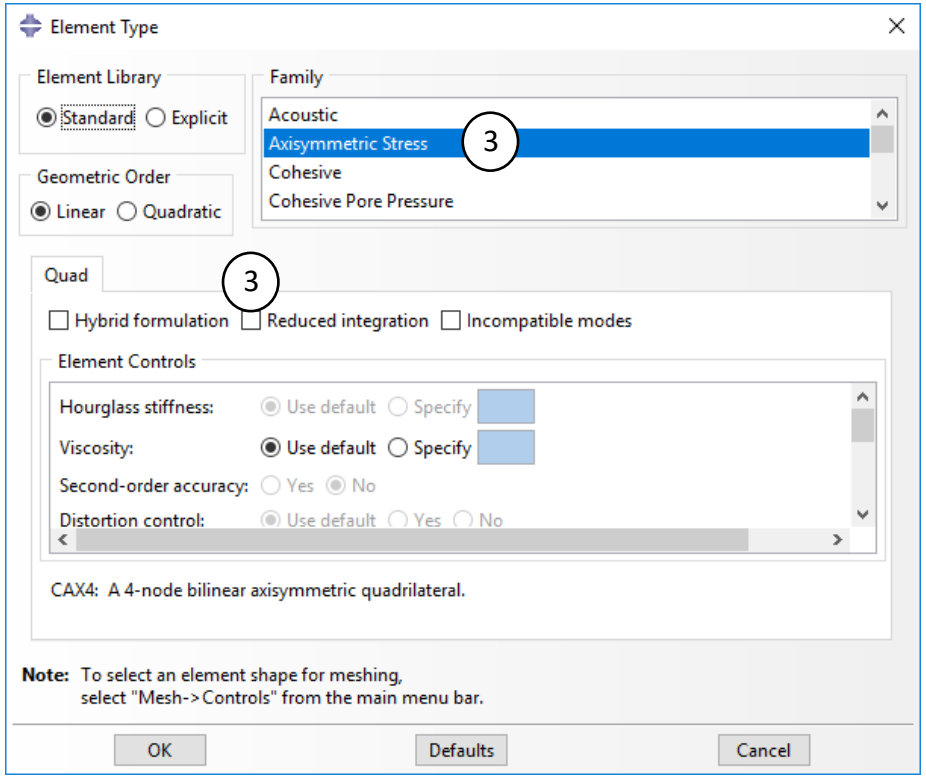


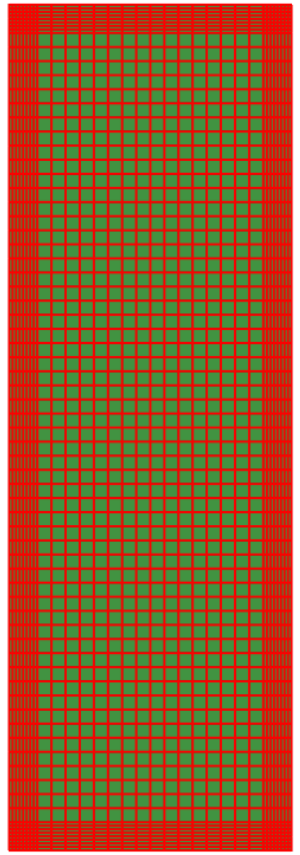
Step 2: Redefining Element Type



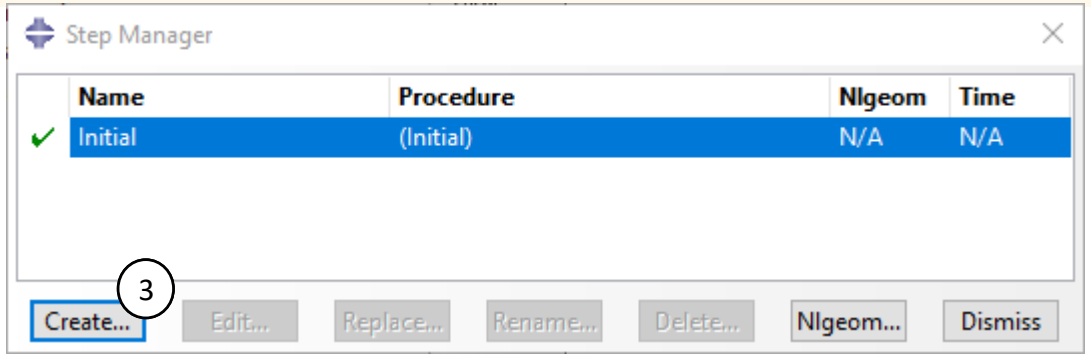
1. Under ***Module***, select ***Mesh*** and ***Part*** for the ***Object*** option
2. In the pulldown menu, go to ***Mesh → Element Type***, box select the entire region then click ***Done***
3. Select ***Axisymmetric Stress*** under ***Family***, uncheck ***Reduced Integration***, leave the rest of the options as default, and click ***OK***.



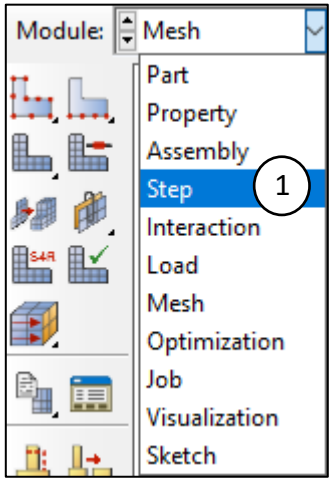


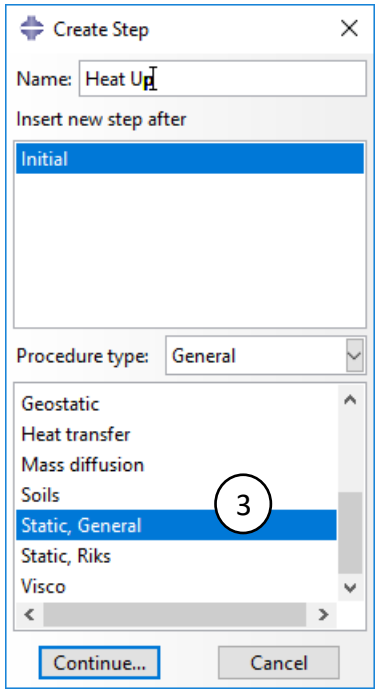


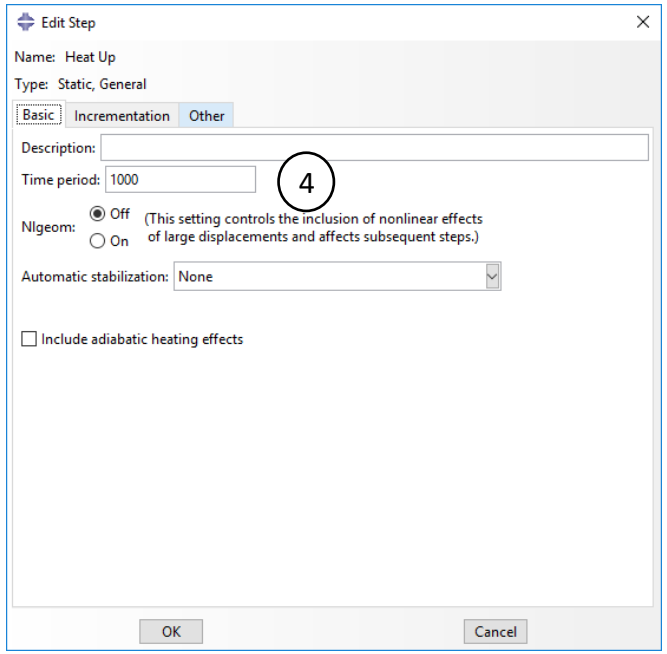
Step 3: Editing The Steps

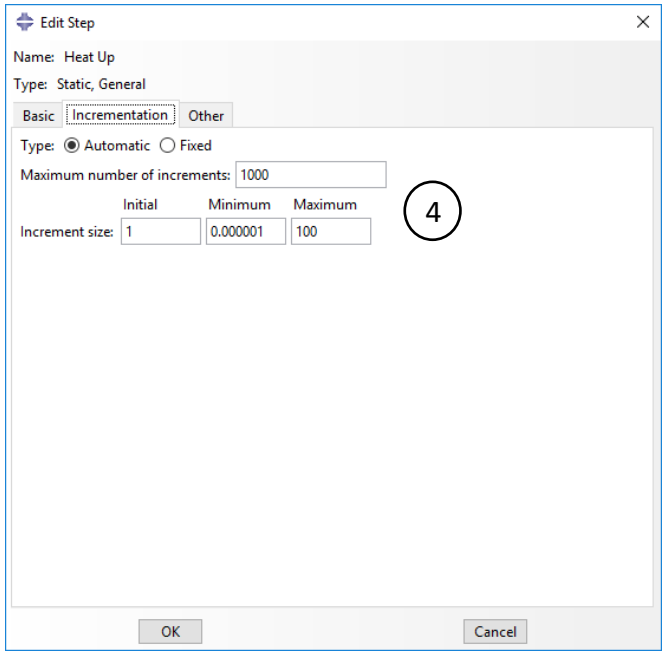


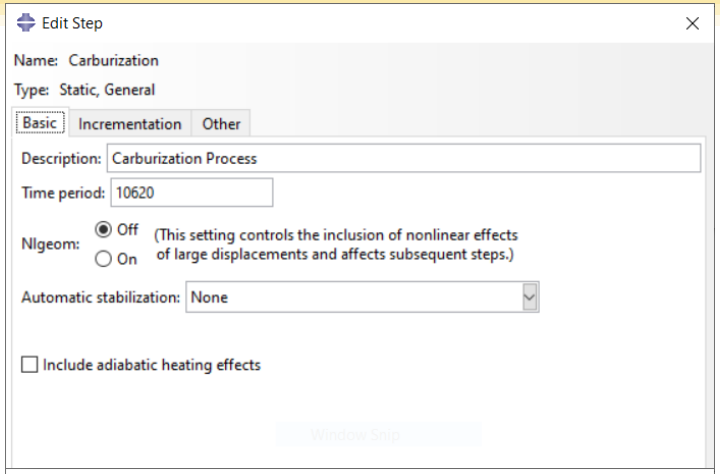
1. Under ***Module***, go to ***Step***
2. In the pulldown menu, select ***Step*** → ***Manager*** and delete all the thermal steps previously defined
3. In ***Step Manager***, click ***Create***, name the step the same as the thermal model, and select ***Static, General***
4. Use the same ***Time Period***, ***Max Increments***, and ***Increment Size*** as the thermal model leaving the rest of the options as default
5. The values for the remaining steps are shown on the next 2 slides

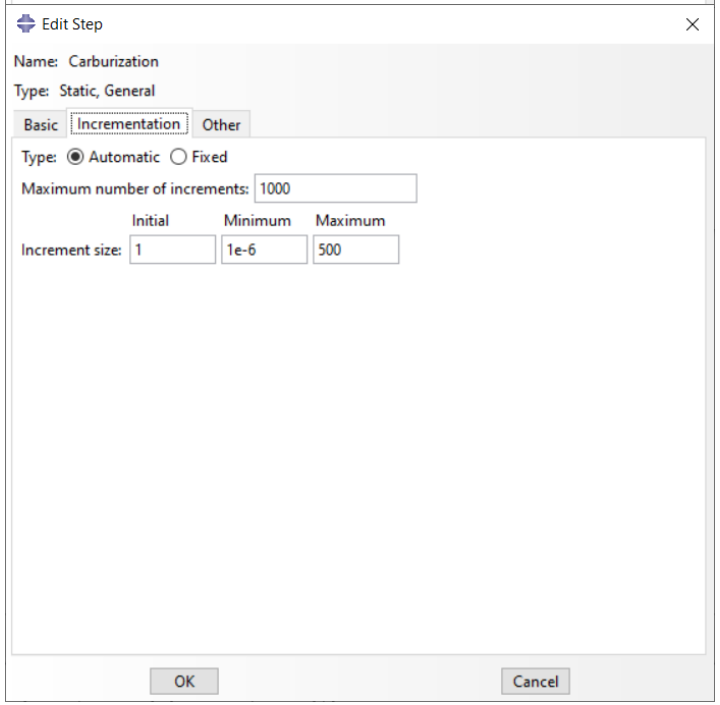


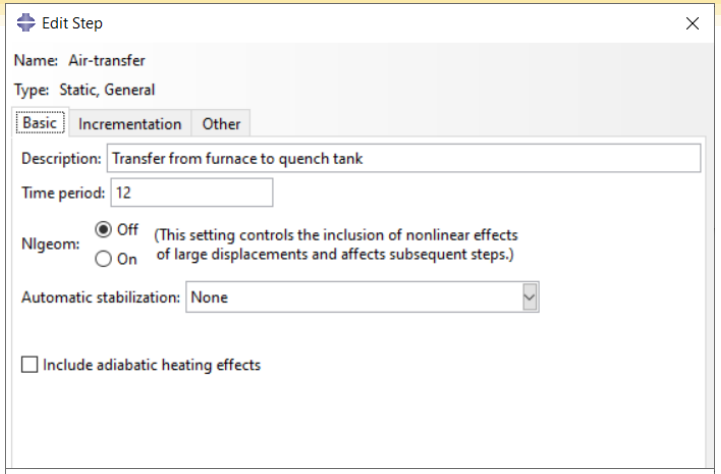


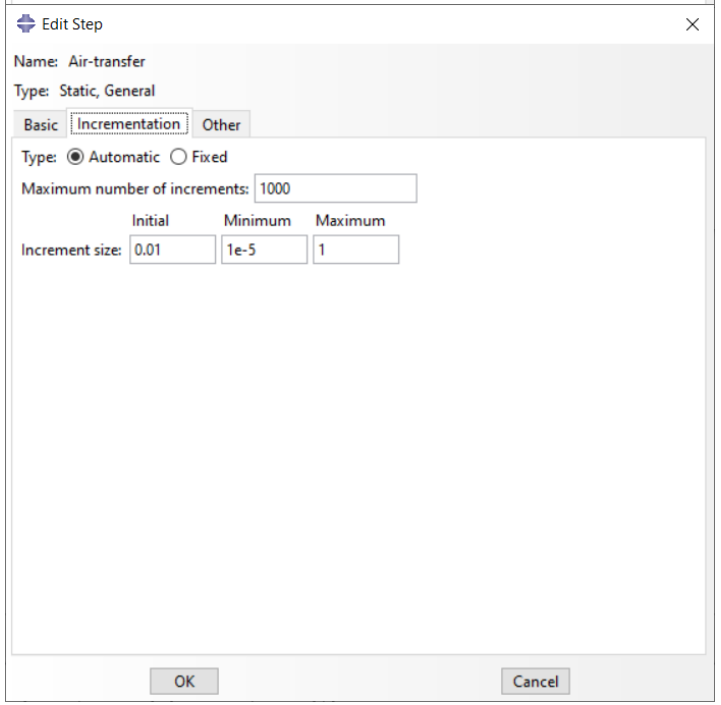


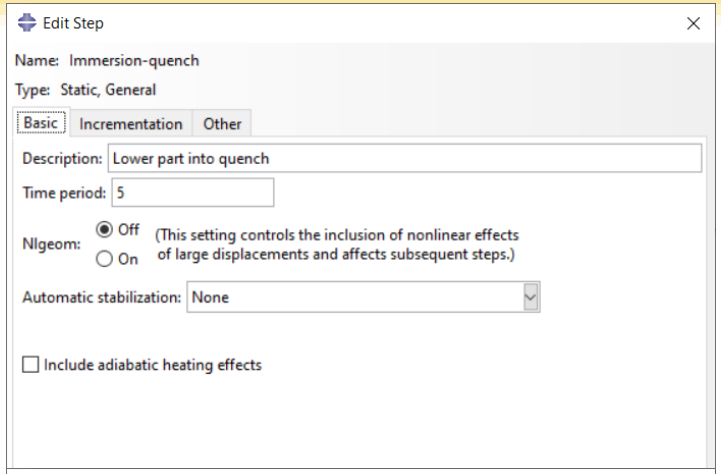


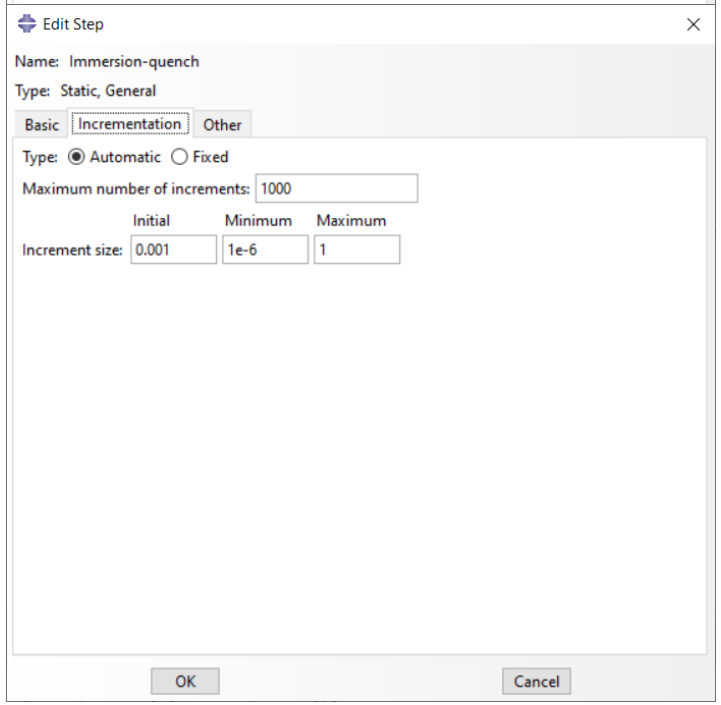


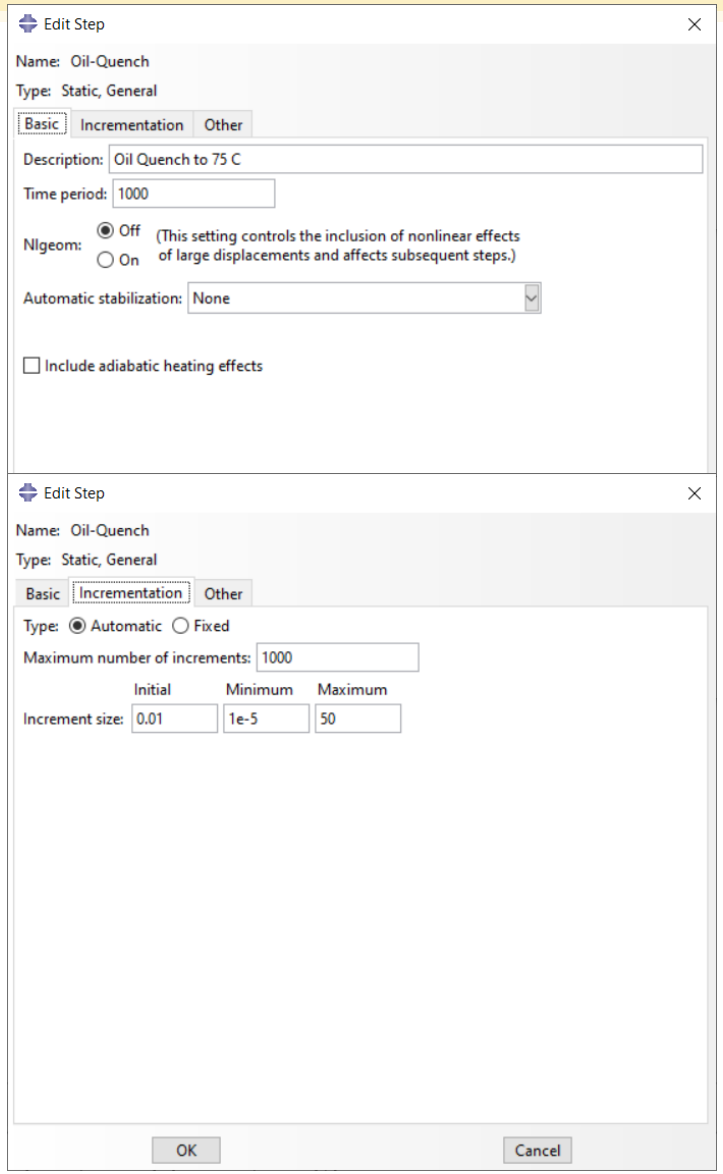


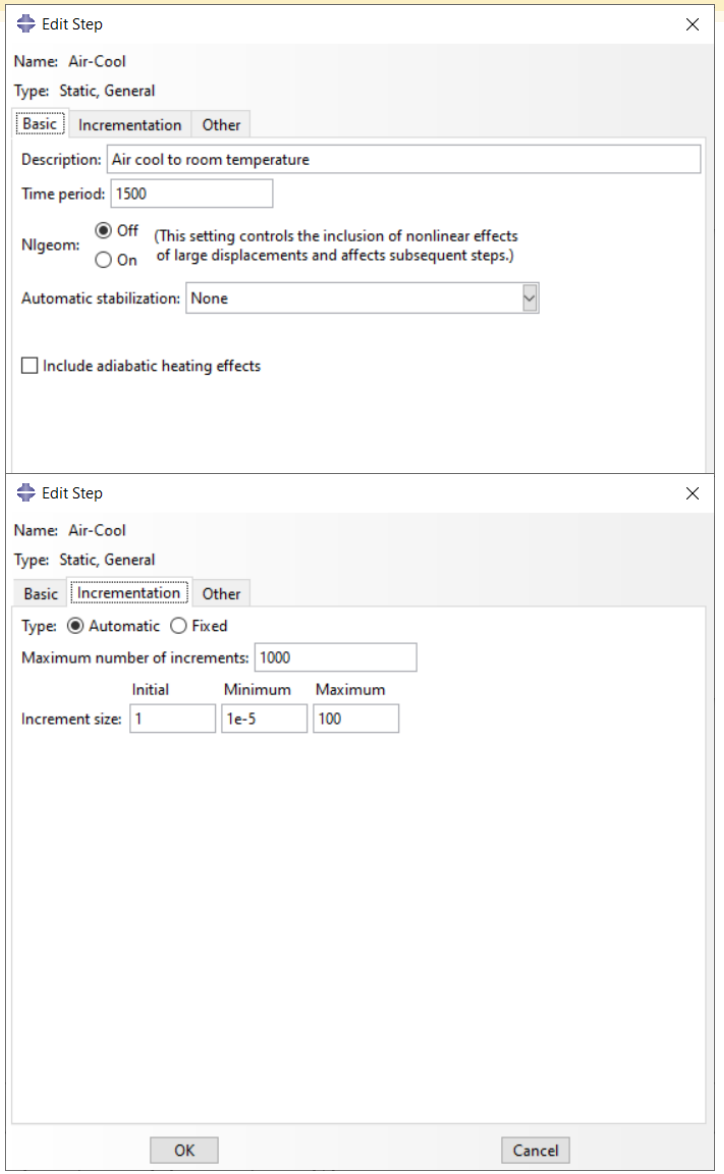




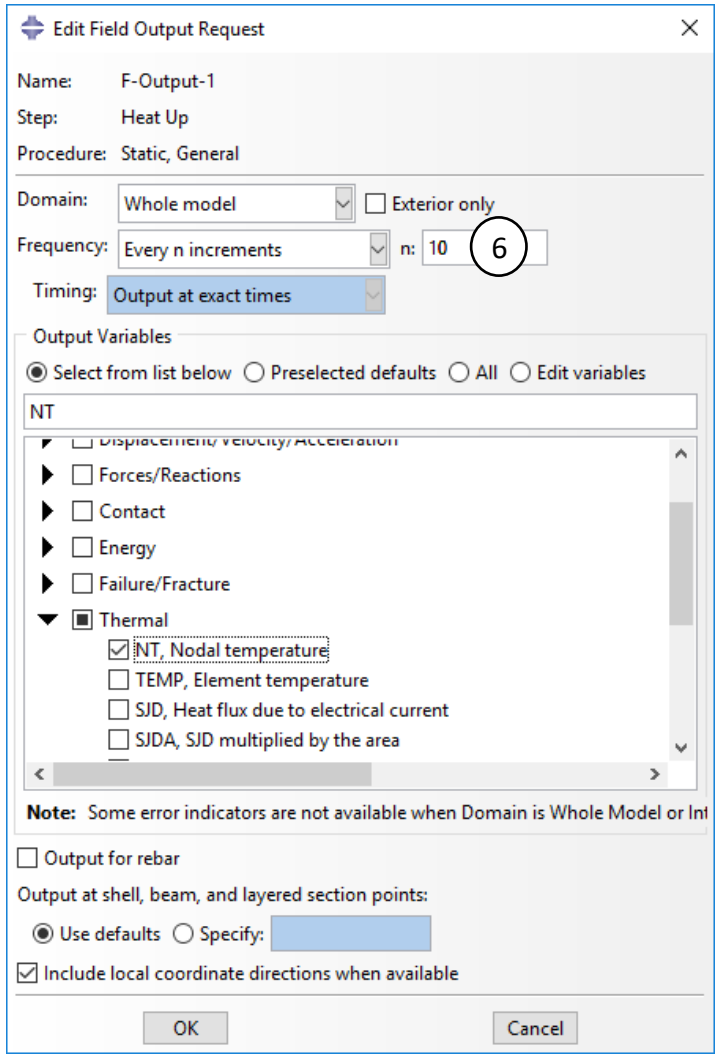


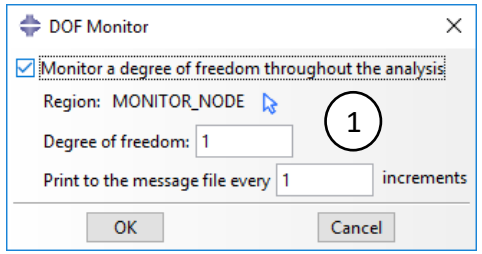




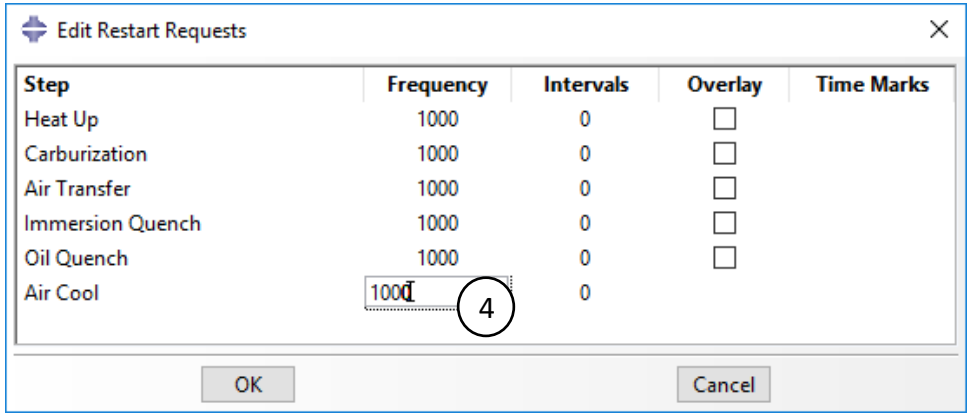


Step 4: Changing Output Definition

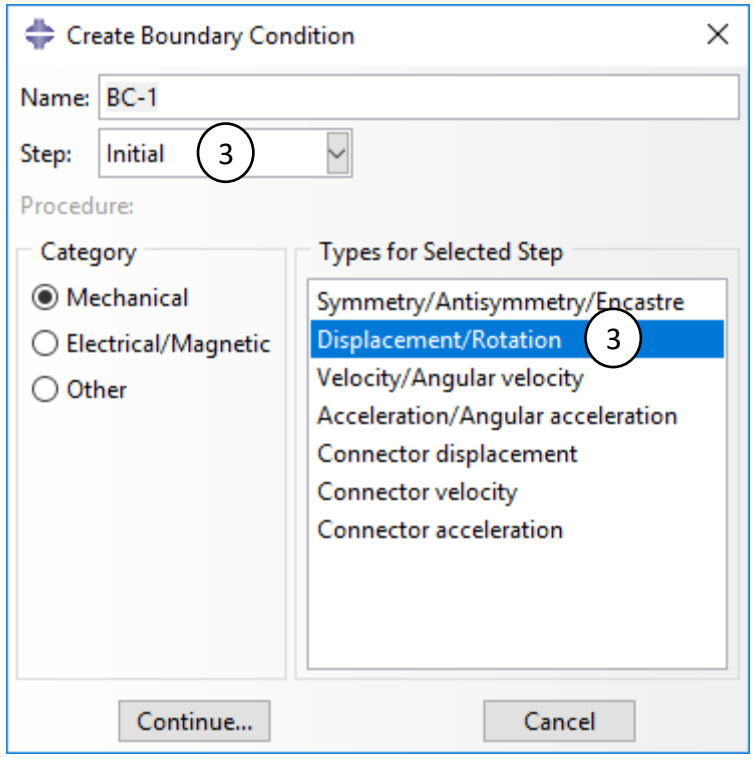


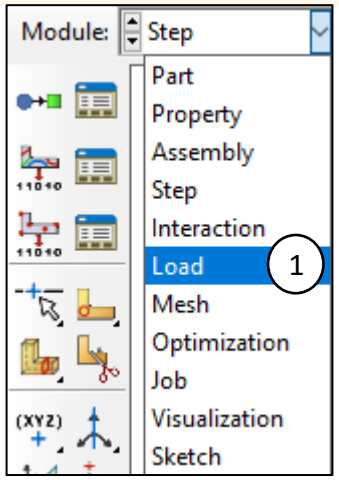


1. In the pulldown menu, select ***Output → DOF Monitor*** and check ***Monitor a degree of freedom throughout the analysis***
2. Select ***Monitor_Node***
3. Change the ***Degree of freedom*** to ***1*** and click ***OK***
4. In the pulldown menu, select ***Output → Restart Requests*** and change the ***Frequency*** for each step to ***1000***
5. Next, in the pulldown menu, select ***Output → Field Output Requests → Manager*** and click ***Edit*** for the ***Heat-Up*** step
6. Change the ***Frequency "n"*** to ***10*** and select the ***NT*** option while leaving the rest of the options as default, and then click ***OK***

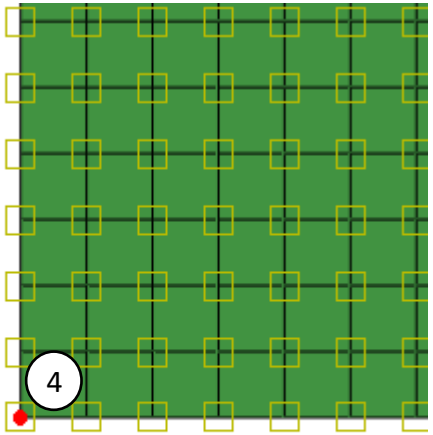


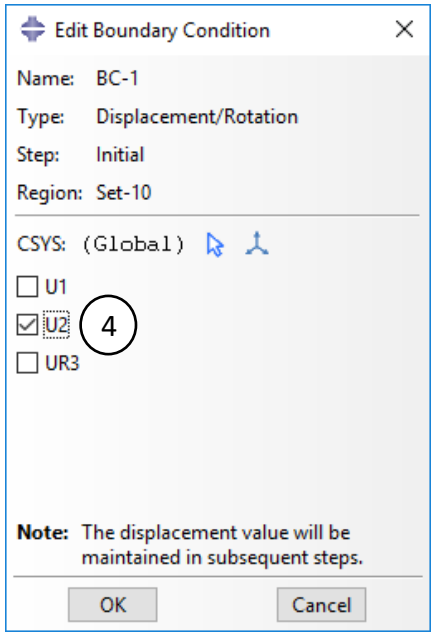
Step 5: Creating Boundary Conditions



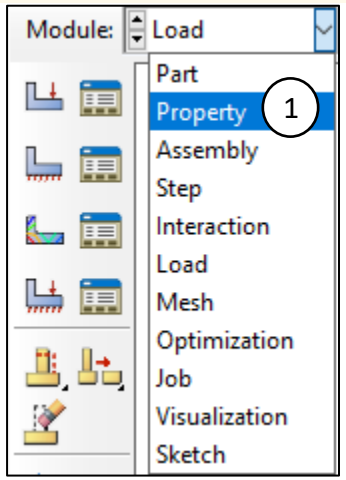


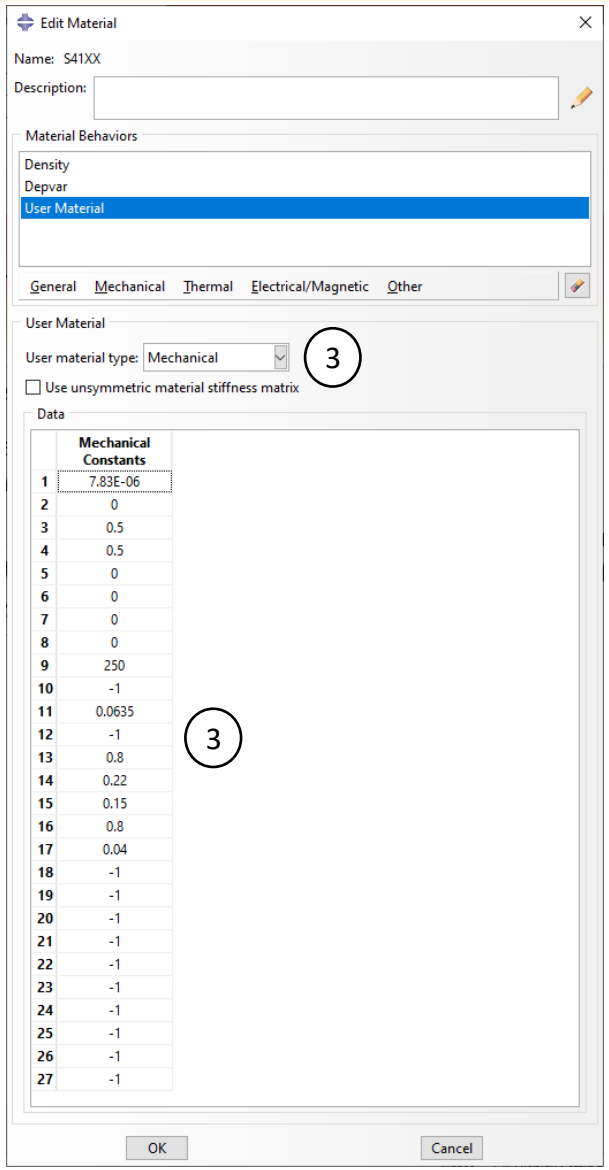
1. Under ***Module***, go to ***Load***
2. In the pulldown menu, select ***BC → Create***
3. Select ***Initial*** inside the ***Step:*** tab, check ***Mechanical*** and select ***Displacement/Rotation,*** then click ***Continue***
4. Choose ***Select in Viewport*** from bottom right-hand side of the screen, select the bottom left node, and click ***Done***
5. Check the ***U2*** option and click ***OK***





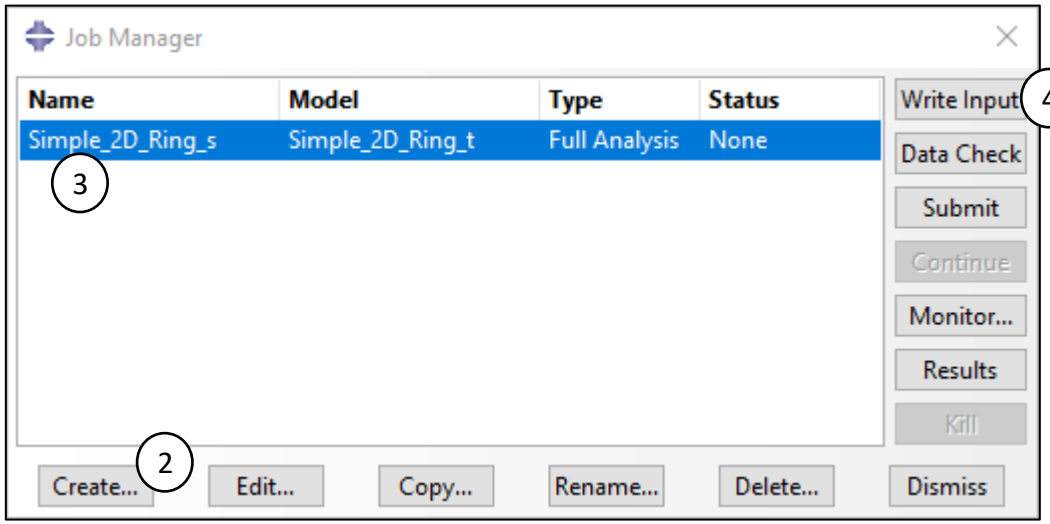
Step 6: Redefining Material Properties





1. Under ***Module***, select ***Property***
2. In the pulldown menu, select ***Material*** → ***Manager*** and click ***Edit*** on the material in the window
3. Select ***User Material***, copy the 27 values currently there for ***Thermal,*** *change* ***User Material Type*** from ***Thermal*** to ***Mechanical***, paste the 27 values into the table, and click ***OK***

Step 7: Writing The Input File



1. Under ***Module***, go to ***Job***
2. In the pulldown menu, select ***Job*** → ***Manager***, and click ***Create***
3. Name the ***Job***, ***Simple_2D_Ring_s*** to represent the stress model and click ***Continue***
4. Use the default options given and write out the input file to the working directory

Step 8: Editing The Input File

```
** MATERIALS
**
*Material, name=S41XX
*Density
 7.83e-06,
*Depvar
    133,
    1,   DEFWF_CARB,          Defined Nascent Carbon Weight Fraction
    2,   HARDNESS,            Total Hardness
    4,   DEFWF_NITROGEN,      Defined Nascent Nitrogen Weight Fraction
    21,  VF_AUSTENITE,        Volume Fraction of Austenite
    34,  VF_FERRITE,          Volume Fraction of Ferrite
    47,  VF_PEARLITE,         Volume Fraction of Pearlite
    60,  VF_UBAINITE,         Volume Fraction of Upper Bainite
    73,  VF_LBAINITE,         Volume Fraction of Lower Bainite
    86,  VF_MARTENSITE,       Volume Fraction of Martensite
    99,  VF_TMARTENSITE,      Volume Fraction of Tempered Martensite
    102, WF_CARB_AUST,        Carbon Weight Fraction in Austenite
    103, WF_CARB_FERR,        Carbon Weight Fraction in Ferrite
    104, WF_CARB_PEARL,       Carbon Weight Fraction in Pearlite
    105, WF_CARB_UBAIN,       Carbon Weight Fraction in Upper Bainite
    106, WF_CARB_LBAIN,       Carbon Weight Fraction in Lower Bainite
    107, WF_CARB_MART,        Carbon Weight Fraction in Martensite
    108, WF_CARB_TMART,       Carbon Weight Fraction in Tempered Martensite
    109, DEFWF_CBDA,          Defined Carbon Weight Fraction in Carbide A
    110, DEF_CBDA_SIZE,       Defined Carbide A Size Factor
    113, DEFWF_NTDA,          Defined Nitrogen Weight Fraction in Nitride A
    114, DEF_NTDA_SIZE,       Defined Nitride A Size Factor
*User Material, constants=27
 7.83e-06,     0.,     0.5,     0.5,     0.,     0.,     0.,     0.
   250.,    -1., 0.0635,    -1.,    0.8,   0.22,   0.15,    0.8
   0.04,    -1.,    -1.,    -1.,    -1.,    -1.,    -1.,    -1.
    -1.,    -1.,    -1.
**
** INTERACTION PROPERTIES
```

1

1. Once the input file has been opened, go to the ***MATERIALS*** section and define the ***Solution-Dependent State Variable (SDV)*** to be analyzed as shown below. Simply copy and paste the text below into the input file.

```
1,   DEFWF_CARB,          Defined Nascent Carbon Weight Fraction
2,   HARDNESS,            Total Hardness
4,   DEFWF_NITROGEN,      Defined Nascent Nitrogen Weight Fraction
21,  VF_AUSTENITE,        Volume Fraction of Austenite
34,  VF_FERRITE,          Volume Fraction of Ferrite
47,  VF_PEARLITE,         Volume Fraction of Pearlite
60,  VF_UBAINITE,         Volume Fraction of Upper Bainite
73,  VF_LBAINITE,         Volume Fraction of Lower Bainite
86,  VF_MARTENSITE,       Volume Fraction of Martensite
99,  VF_TMARTENSITE,      Volume Fraction of Tempered Martensite
102, WF_CARB_AUST,        Carbon Weight Fraction in Austenite
103, WF_CARB_FERR,        Carbon Weight Fraction in Ferrite
104, WF_CARB_PEARL,       Carbon Weight Fraction in Pearlite
105, WF_CARB_UBAIN,       Carbon Weight Fraction in Upper Bainite
106, WF_CARB_LBAIN,       Carbon Weight Fraction in Lower Bainite
107, WF_CARB_MART,        Carbon Weight Fraction in Martensite
108, WF_CARB_TMART,       Carbon Weight Fraction in Tempered Martensite
109, DEFWF_CBDA,          Defined Carbon Weight Fraction in Carbide A
110, DEF_CBDA_SIZE,       Defined Carbide A Size Factor
113, DEFWF_NTDA,          Defined Nitrogen Weight Fraction in Nitride A
114, DEF_NTDA_SIZE,       Defined Nitride A Size Factor
```

Step 8b: Editing The Input File

```
*Depvar
    133,
    1,   DEFWF_CARB,          Defined Nascent Carbon Weight Fraction
    2,   HARDNESS,            Total Hardness
    4,   DEFWF_NITROGEN,      Defined Nascent Nitrogen Weight Fraction
    21,  VF_AUSTENITE,        Volume Fraction of Austenite
    34,  VF_FERRITE,          Volume Fraction of Ferrite
    47,  VF_PEARLITE,         Volume Fraction of Pearlite
    60,  VF_UBAINITE,         Volume Fraction of Upper Bainite
    73,  VF_LBAINITE,         Volume Fraction of Lower Bainite
    86,  VF_MARTENSITE,       Volume Fraction of Martensite
    99,  VF_TMARTENSITE,      Volume Fraction of Tempered Martensite
    102, WF_CARB_AUST,        Carbon Weight Fraction in Austenite
    103, WF_CARB_FERR,        Carbon Weight Fraction in Ferrite
    104, WF_CARB_PEARL,       Carbon Weight Fraction in Pearlite
    105, WF_CARB_UBAIN,       Carbon Weight Fraction in Upper Bainite
    106, WF_CARB_LBAIN,       Carbon Weight Fraction in Lower Bainite
    107, WF_CARB_MART,        Carbon Weight Fraction in Martensite
    108, WF_CARB_TMART,       Carbon Weight Fraction in Tempered Martensite
    109, DEFWF_CBDA,          Defined Carbon Weight Fraction in Carbide A
    110, DEF_CBDA_SIZE,       Defined Carbide A Size Factor
    113, DEFWF_NTDA,          Defined Nitrogen Weight Fraction in Nitride A
    114, DEF_NTDA_SIZE,       Defined Nitride A Size Factor
*User Material, constants=27, Type=Mechanical
 7.83e-06,     0.,    0.5,    0.5,     0.,      0.,     0.,     0.
   250.,    -1., 0.0635,    -1.,    0.8,   0.22,   0.15,    0.8
   0.04,    -1.,    -1.,    -1.,    -1.,    -1.,    -1.,    -1.
    -1.,    -1.,    -1.
**
** INTERACTION PROPERTIES
```

1

1. Next, Define the ***type*** to be ***Mechanical*** in the ****User Material*** definition as seen in the bottom left image if it has not been done so already
2. In the ***Step*** section underneath the first step (***Heat Up***), input the lines seen below to help with convergence

```
4874 ** STEP: Heat Up
4875 **
4876 *Step, name="Heat Up", nlgeom=NO, inc=1000
4877 *Static
4878 1., 1000., 1e-06, 100.
4879 **
4880 *CONTROLS, PARAMETERS=LINE SEARCH
4881 6,
4882 *CONTROLS, PARAMETERS=TIME INCREMENTATION
4883 20, 30
4884 *CONTROLS, FIELD=DISPLACEMENT, PARAMETERS=FIELD
4885 0.05,0.05,
4886 **
4887 ** OUTPUT REQUESTS
```

2

Step 8b: Editing The Input File

```
4874 ** STEP: Heat Up
4875 **
4876 *Step, name="Heat Up", nlgeom=NO, inc=1000
4877 *Static
4878 1., 1000., 1e-06, 100.
4879 **
4880 *CONTROLS, PARAMETERS=LINE SEARCH
4881 6,
4882 *CONTROLS, PARAMETERS=TIME INCREMENTATION
4883 20, 30
4884 *CONTROLS, FIELD=DISPLACEMENT, PARAMETERS=FIELD
4885 0.05,0.05,
4886 **
4887 *TEMPERATURE, FILE=Simple_2D_Ring_t, BSTEP=1, ESTEP=1
4888 ** OUTPUT REQUESTS
4889 **
4890 *Restart, write, frequency=0
4891 **
4892 ** FIELD OUTPUT: F-Output-1
4893 **
4894 *Output, field, variable=PRESELECT
4895 **
4896 ** HISTORY OUTPUT: H-Output-1
4897 **
4898 *Output, history, variable=PRESELECT
4899 *End Step
```

1

1. In each step, add the command

TEMPERATURE, FILE=ThermalFileName, BSTEP=Step#, ESTEP=Step#

Where # represents the step number 1- 6

This reads in the temperature history from the thermal model

2. In the ***Carburization*** step, also add the command

FIELD, OP=NEW, VAR=3, INPUT=NameOfCarbonContentFile.nod

- This inputs the carbon content of the last step during the carburization model into the stress model

```
4901 **
4902 ** STEP: Carburization
4903 **
4904 *Step, name=Carburization, nlgeom=NO, inc=1000
4905 *Static
4906 1., 10620., 1e-06, 500.
4907 **
4908 *FIELD, OP=NEW, VAR=3, INPUT=Simple_Ring_cc.nod
4909 ** OUTPUT REQUESTS
4910 **
4911 *Restart, write, frequency=0
4912 **
4913 ** FIELD OUTPUT: F-Output-1
4914 **
4915 *Output, field, variable=PRESELECT
4916 **
4917 ** HISTORY OUTPUT: H-Output-1
4918 **
4919 *Output, history, variable=PRESELECT
4920 *End Step
```

2

Step 8c: Editing The Input File

```
4874 ** STEP: Heat Up
4875 **
4876 *Step, name="Heat Up", nlgeom=NO, inc=1000, amp=step
4877 *Static
4878 1., 1000., 1e-06, 100.
4879 **
```

(1)

```
4902 ** STEP: Carburization
4903 **
4904 *Step, name=Carburization, nlgeom=NO, inc=1000, amp=ramp
4905 *Static
4906 1., 10620., 1e-06, 500.
4907 **
```

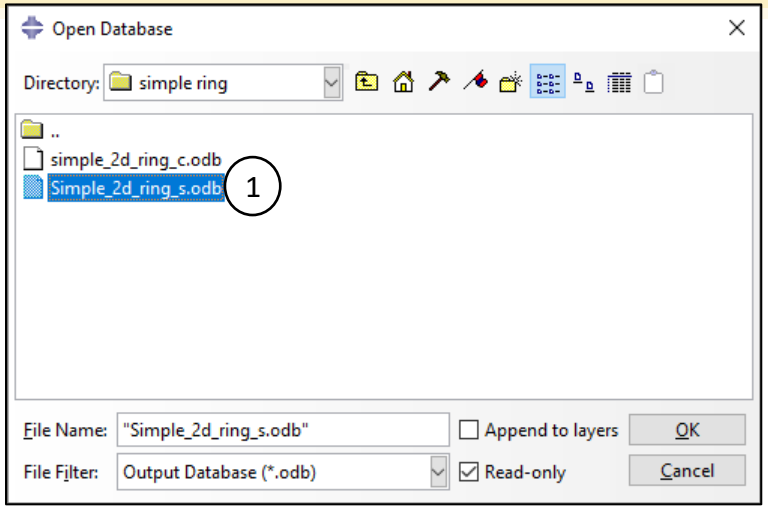
(1)

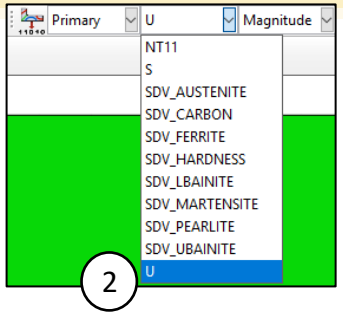
```
*Restart, write, frequency=1000
*Monitor, dof=1, node=A_MONITOR_NODE, frequency=1
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, frequency=10
*Node Output
NT,U
*Element Output, directions=YES
S,SDV1, SDV2, SDV4, SDV21, SDV34, SDV47, SDV60, SDV73, SDV86, SDV99, SDV102, SDV103
SDV104, SDV105, SDV106, SDV107, SDV108, SDV109, SDV110, SDV113, SDV114
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
```

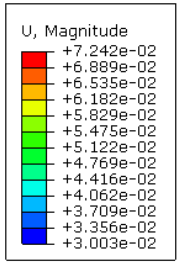
(2)

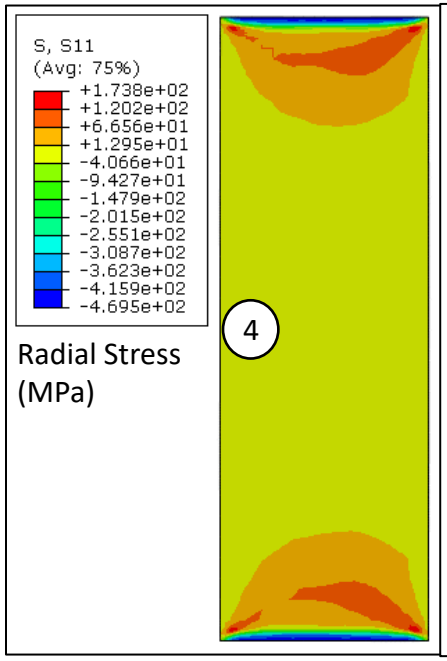
1. In each ****Step*** command, write in ***amp=step*** at the end except for the ***Carburization*** step where ***amp=ramp*** will be added
2. Define the variables to be output for the analysis by writing out the ****Node Output*** and ****Element Output*** command before ****End Step*** for each ***Step***. This can simply be copy and pasted from the thermal model input file, and a ***U*** added to ****Node Output*** and ***S*** added to ****Element Output***. The ****Node Output***, ****Element Output***, and ****Contact Output*** currently in the input file can be overwritten
3. The model can now be executed using ***Abaqus Command***, just as the carburization and thermal models

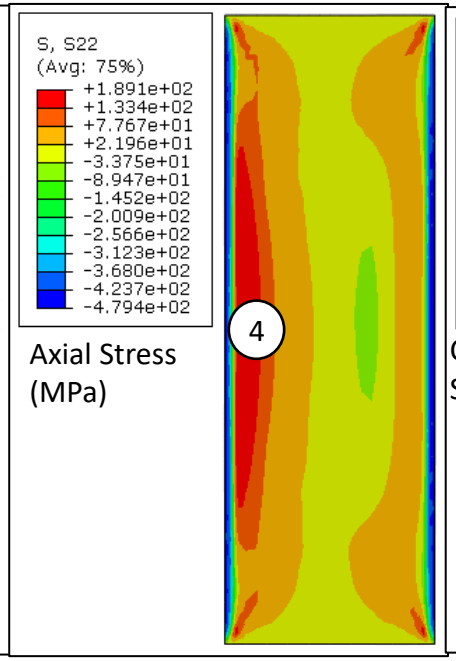
Step 9: Post Process - Stress Results

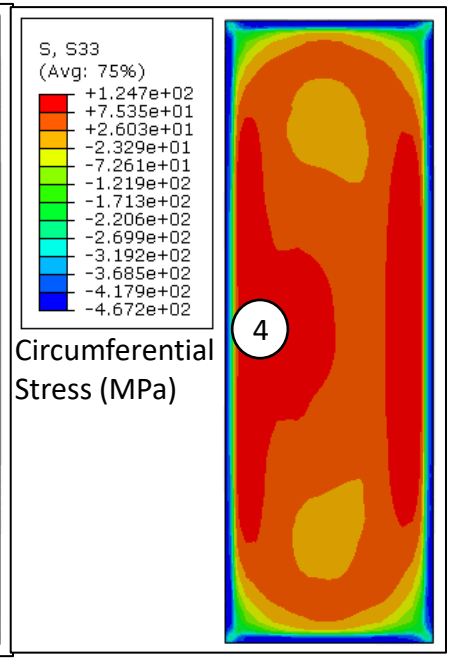












1. To open the file, go back to ***Abaqus Cae***, under ***Module***, select ***Visualization*** and open the .odb file located in the working directory
2. As with all simulations, take a moment and check the results to ensure that they make sense. We expect displacements(U) to be small, somewhere on the order of .1mm or 100μm. In the field drop-down menu, select U for displacement
3. The contour will show displacement results from the fixed reference node. In this case, the bottom left corner(inside edge of the part) is the reference for all displacement.
4. In the field drop-down menu selecting S for stress allows the user to show several different types of stress calculations on the contour; Such as vonMises, min and max principal stresses, hydrostatic pressure(where compression is positive), and even directional stress in the S11(x), S22(y), and S33(z) directions. For axisymmetric models such as this one, S11 represents radial stress, S22 represents axial stress, and S33 represents circumferential stress
5. Path plots and history plots for these field variables, as well as the DANTE solutions variables can be obtained using the same methods described above in the thermal and carburization sections