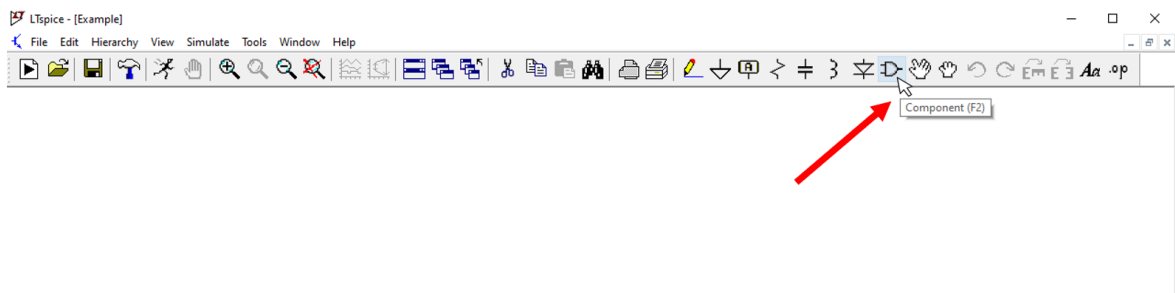


# Using the LTspice Magnetic Core Model

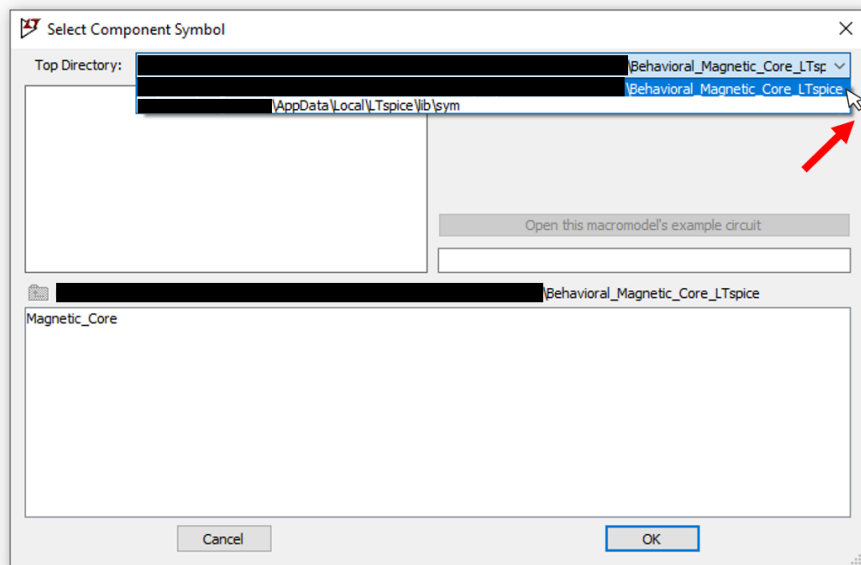
1. Create new schematic in LTspice in the same folder as the magnetic core schematic and symbol.

## Inserting Model:

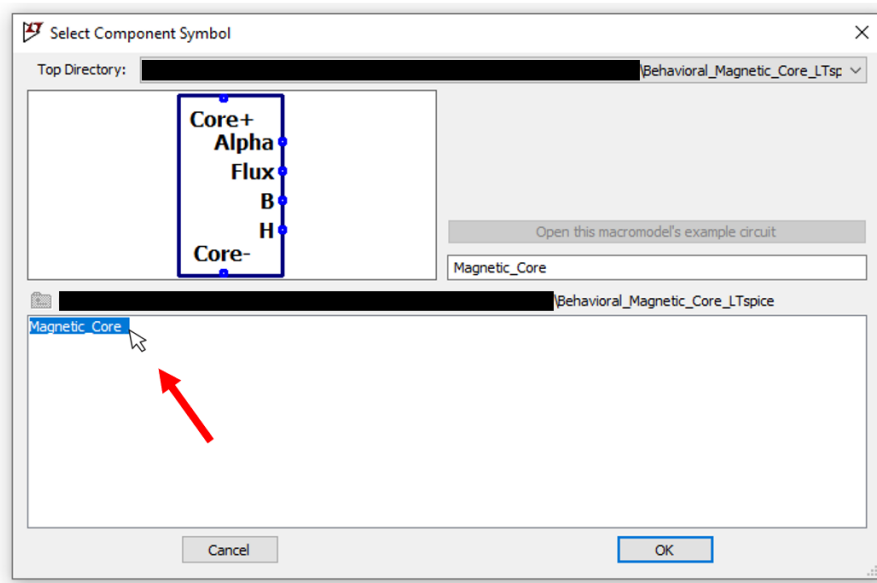
2. Add Component:



3. Select the directory of the magnetic core schematic and symbol

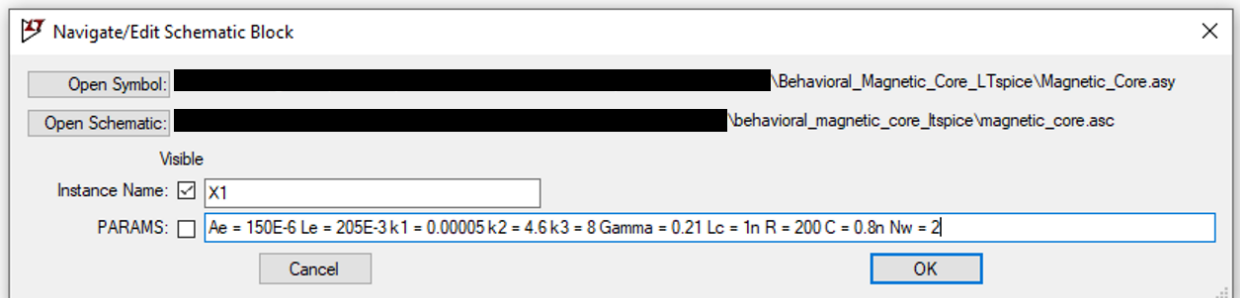


#### 4. Select Magnetic\_Core



#### 5. Right click on the magnetic core symbol in your circuit then input the following Parameters:

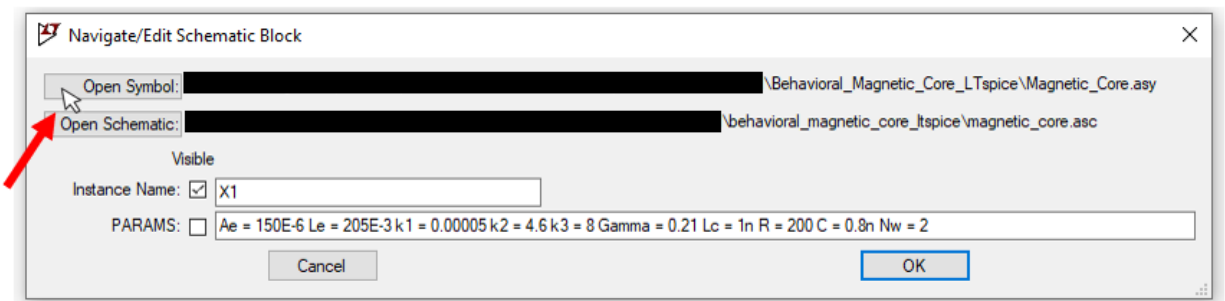
$A_e$ ,  $L_e$ ,  $k_1$ ,  $k_2$ ,  $k_3$ ,  $\Gamma$ ,  $L_c$ ,  $R$ ,  $C$ ,  $N_w$



# Troubleshooting

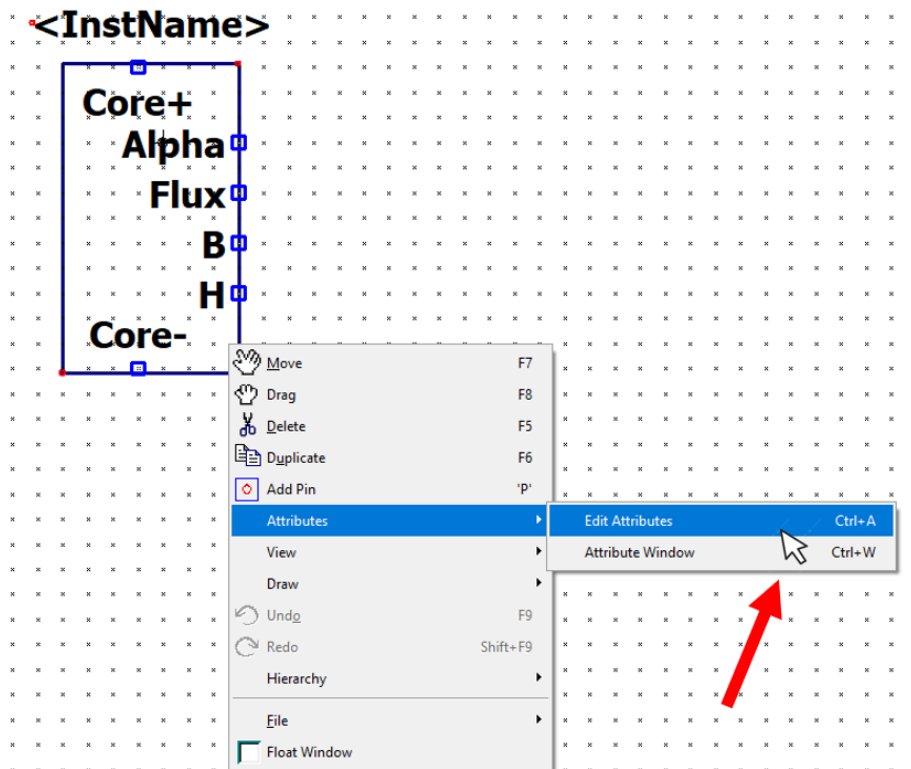
Must ensure the Value field of the Symbol Attribute Editor has the same name as the magnetic core circuit (“Magnetic\_Core”)

Right click on the magnetic core symbol in your circuit



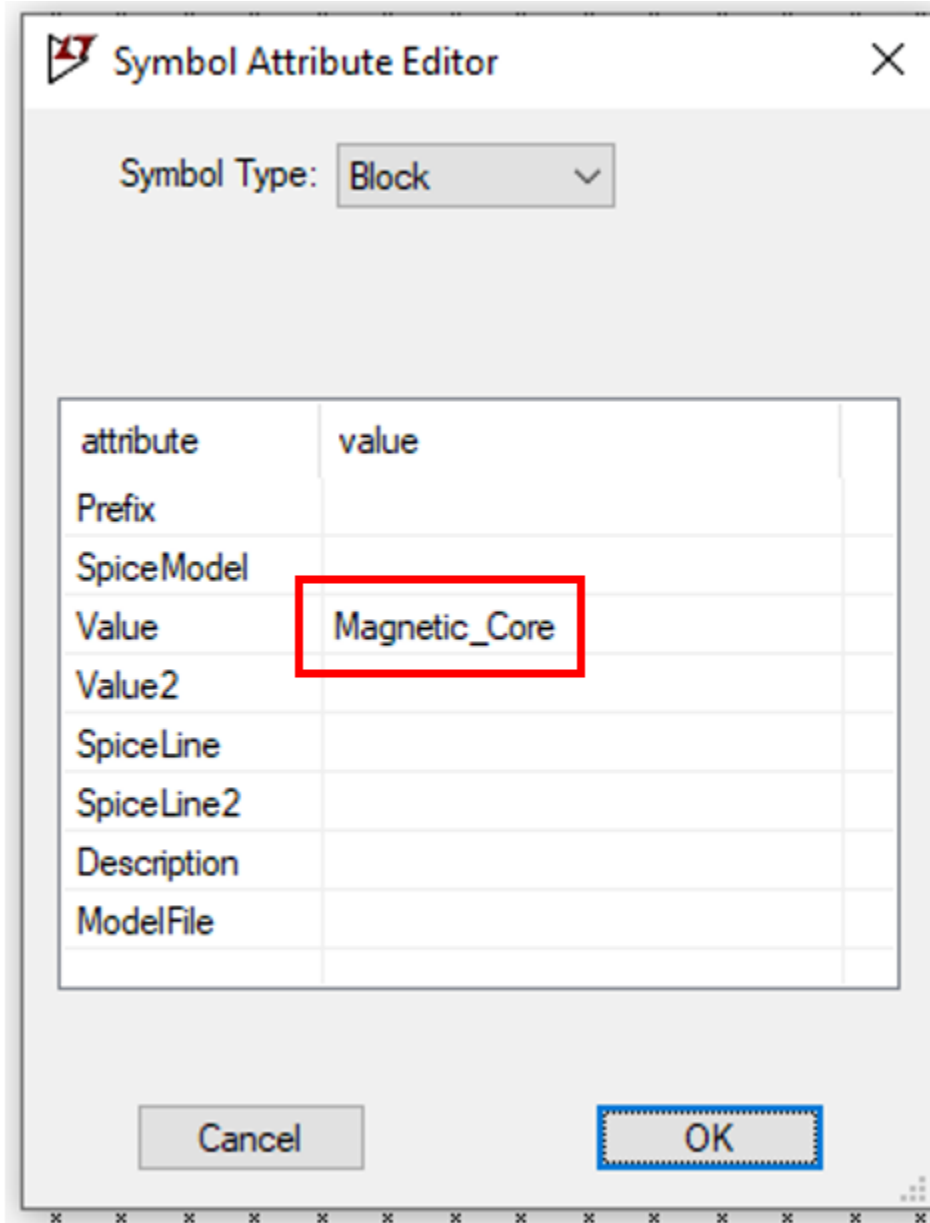
Select open symbol

Right click on the symbol in the symbol editor



Select Edit Attributes

Enter the file name of the magnetic core model circuit in the Value field.



The image shows a 'Symbol Attribute Editor' dialog box. At the top, there is a title bar with a red lightning bolt icon and the text 'Symbol Attribute Editor'. Below the title bar, there is a 'Symbol Type:' label followed by a dropdown menu showing 'Block'. The main area of the dialog is a table with two columns: 'attribute' and 'value'. The table has several rows, with the 'Value' row highlighted by a red rectangular box. The 'Value' row contains the text 'Magnetic\_Core'. At the bottom of the dialog, there are two buttons: 'Cancel' and 'OK'. The 'OK' button is highlighted with a blue dashed border.

attribute	value
Prefix	
SpiceModel	
Value	Magnetic_Core
Value2	
SpiceLine	
SpiceLine2	
Description	
ModelFile	