

Procedure for making new components using existing Multisim Symbols

1. Tools (Alt + t)
2. Database (d),
3. Database manager (m)
4. Select similar component type from your User, Master, or Corporate Database
5. Edit (e)
6. Select "General" tab on "Component Properties"
7. Enter whatever Name you want for the component (necessary) as well as Date, Author, and Function (optional)
8. Select "Model" tab on "Component Properties"
9. Add/Edit (e)
10. Add (a) on "Select a Model"
11. Enter whatever "Model ID" you wish to use for the component's SPICE model in "Set Parameters"
12. Click "OK"
13. Load model from file (o) on "Select a Model"
14. Find and select your SPICE model (unless you change the model's file extension from ".txt" to ".cir", you'll need to select "All files (*.*)" in the lower right dropdown box to find and select it)
15. Select your model and Open (Enter)
16. Save (s) on "Select a Model"
17. Click "Select" on "Select a Model"
18. Verify/correct the "Symbol to model pin mapping" on "Component Properties"
19. Select the old (from step 4.) "Model name" on "Component Properties"
20. Delete a model (d) on "Component Properties"
21. If other sections of the component use the same model click "Copy to..." on "Component Properties"
 - a. On "Select Target" click in appropriate (i.e. "Section B") Box
 - b. Click OK (Enter) on "Select Target"
 - c. Repeat steps 17 - 20
22. Ok (o or Enter) on "Component Properties"
23. On "Select Destination Family Name" either one or add a new one for your component and its SPICE model
24. OK (o or Enter) on "Select Destination Family Name"
25. Close (c) "Database Manager" unless you want to build another component