



ICS SPICE MODELS

APPLICATION NOTE

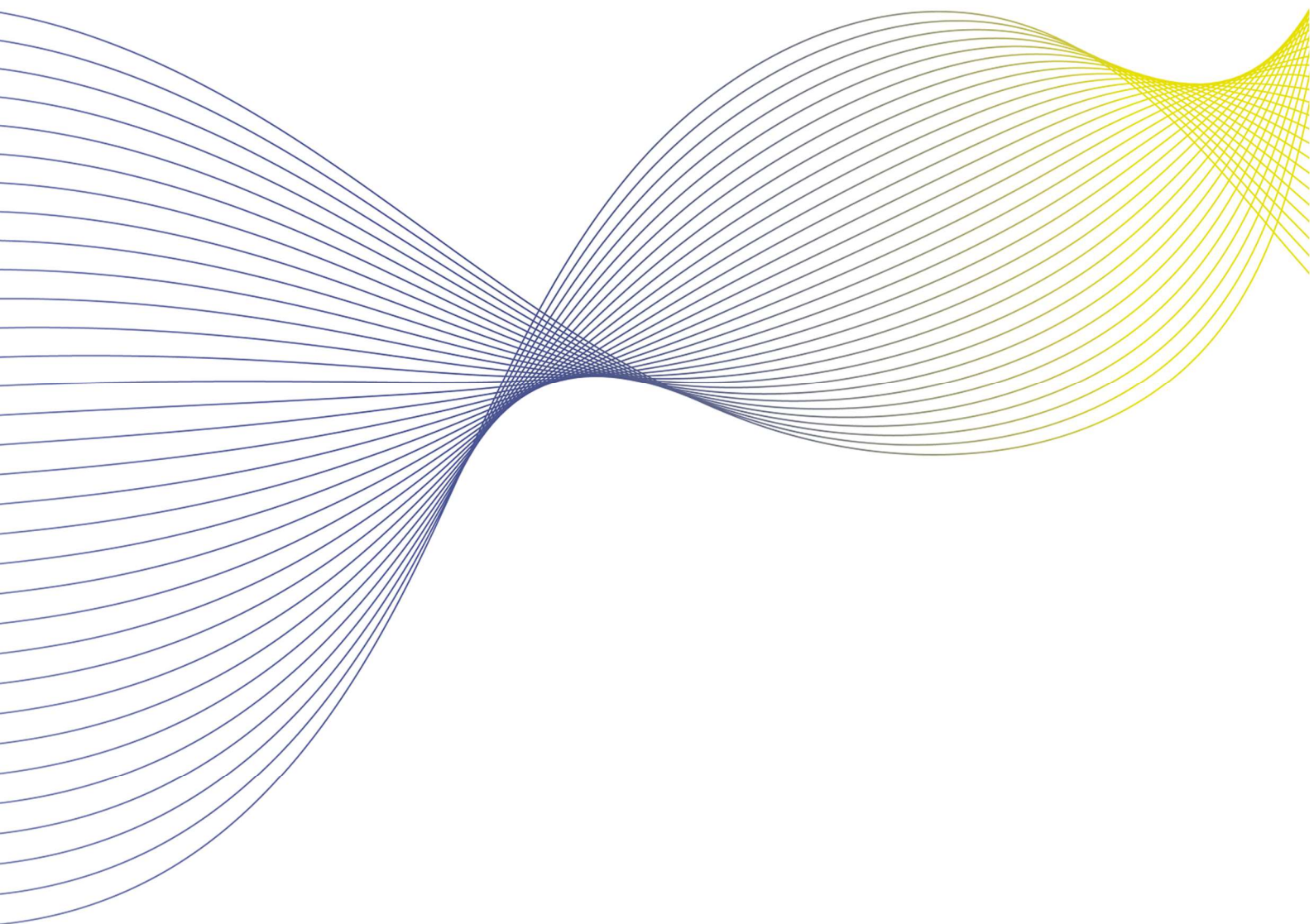


TABLE OF CONTENTS

Introduction.....	2
Procedure for Using library Folder in LTSpice	2
Procedure for Using Application schematic in LTSpice	3



INTRODUCTION

Behavioural models for LEM current sensors were created for simulation in LTSpice. The document details the process of importing the device symbol into LTSpice, adjusting the model's sensitivity and other specific parameters, and setting up a basic test bench to evaluate the model's performance.

Revision	Date	Comment	Responsible
1.0	Feb 2025	Initial Release – HMSR_5V	Murthy Batchu (mbt@lem.com)

PROCEDURE FOR USING LIBRARY FOLDER IN LTSPICE

1. Download and Extract Files:

- Download the zip folder LEM_ICS_SpiceLibrary.zip
- Extract the downloaded files to a known location on your computer.

2. Organize Files and Copy to LTSpice Directory:

- Inside the extracted LEM_ICS_SpiceLibrary.zip folder, there are three subfolders: LEMsub, LEMsym, LEMAppSch.
- Navigate to the LTspice directory, typically located in the Documents section or Local folders of your PC (for ex: C:\Users**USERNAME**\AppData\Local\LTspice\lib).
- Place downloaded symbol files folder (LEMsym) in the LTspice symbol files folder (C:\Users**USERNAME** \AppData\Local\LTspice\lib\sym) and the library files under LTspice library files folder(C:\Users\ **USERNAME** \AppData\Local\LTspice\lib\sub)..

3. Configure LTSpice (optional):

- It is also possible to copy the downloaded files into any other folder of your choice and provide the search path for the symbols and library search paths.
- Open LTSpice.
- Go to Tools > Control Panel > Sym. & Lib. Search Paths.
- Add the path to the LEM_Library folder to the search paths for both symbols and libraries.
- In the schematic, right-click and select Draft > SPICE Directive or press “.” (shortcut is based on LTspice 24.0.12, it could vary based on the version one has).

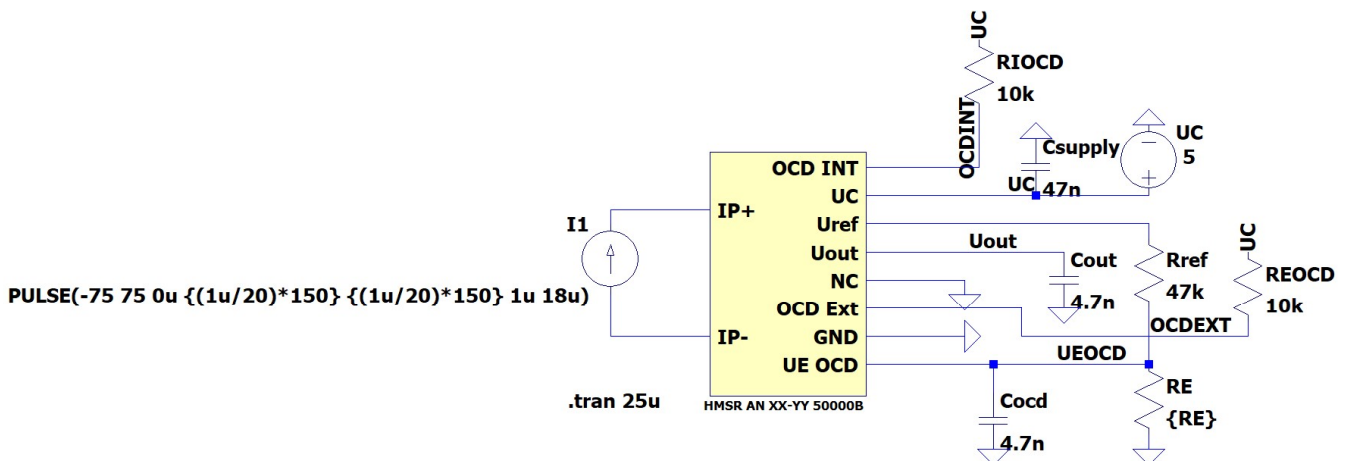


- In the SPICE Directive window, type `.include .../LEM_Library/lib/your_model.lib` and place it on the schematic.

PROCEDURE FOR USING APPLICATION SCHEMATIC IN LTSPICE

1. Organize Files and apply setting required:

- LEM_ICS_SpiceLibrary.zip folder contains the folder LEMAppSch where the application schematics corresponding to each of the spice model are stored.



`.lib HMSRAN.lib`

`.param Uref=2.5 ;5V version`

`.param Ithreshold=1 ; External OCD threshold, Multiple of FSR`

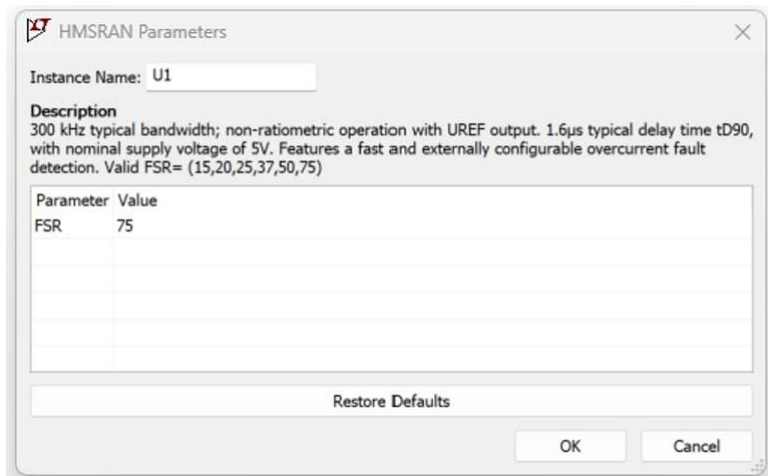
`.param Ueocd={ (7.8125-Ithreshold)/6.5 } ; Required Ueocd for the set OCD threshold`

`.param RE={ Ueocd*47k/(Uref-Ueocd) } ; RE value for setting the required external OCD.`

These settings can be used to define the value of RE based on Ithreshold value here. Alternatively, RE value can be directly set to define external overcurrent threshold.

- Double click the symbol to update the relevant parameters like FSR etc., with the valid values described in the popup.





- Change the other settings like the current source (I1) etc., based on the need and run the simulation (ALT+R).
- After the successful simulation run, LTspice opens the corresponding plot window. Add the required traces in the plot window to verify the results.

