

LTspice Model

Rectifier Diode

Taiwan Semiconductor

HS1G-TR

Model Information

Model	A macro model	
Call Name	MDC_HS1G-TR_LT	
Pin Assign	1:Port1 2:Port2	
File List	Model Library	MDC_HS1G-TR_LT.lib
	Model Report	MDC_HS1G-TR_LT.pdf(this file)
Verified Simulator Version	LTspice XVII	
Note		

References

The information which was used for modeling is as follow:

[Data Sheet]

●Date/Version	Version: L2102
●Product name	HS1G-TR
●Company name	Taiwan Semiconductor

[Characteristics listed]

●Characteristics	Junction Capacitor – Reverse Voltage
	Reverse Current – Reverse Voltage
	Forward Current – Forward Voltage
	Reverse Recovery Time

Simulation Condition

This table shows the range of evaluated simulation range that was not occurs any convergence problems in this area.

Item	Condition	Unit
Temperature	25,125	deg C

○ : Implemented
× : Not Implemented
— : Not applicable

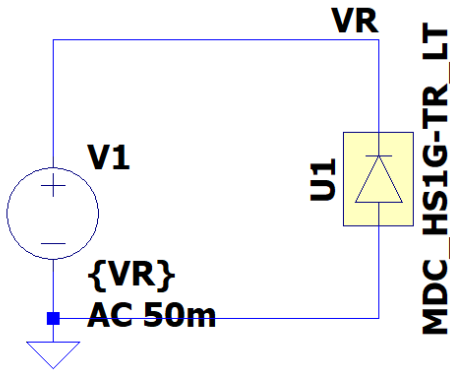
Model Functions Table
RANK=1

Functions	RANK	Implemented
IF-VF	1	○
IR-VR(Temp)	1	○
Capacitance	1	○
Reverse recovery characteristics	1	○
Rectification characteristics(Bridge)	1	—
Surge voltage-Transient	2	—

Junction Capacitor – Reverse Voltage

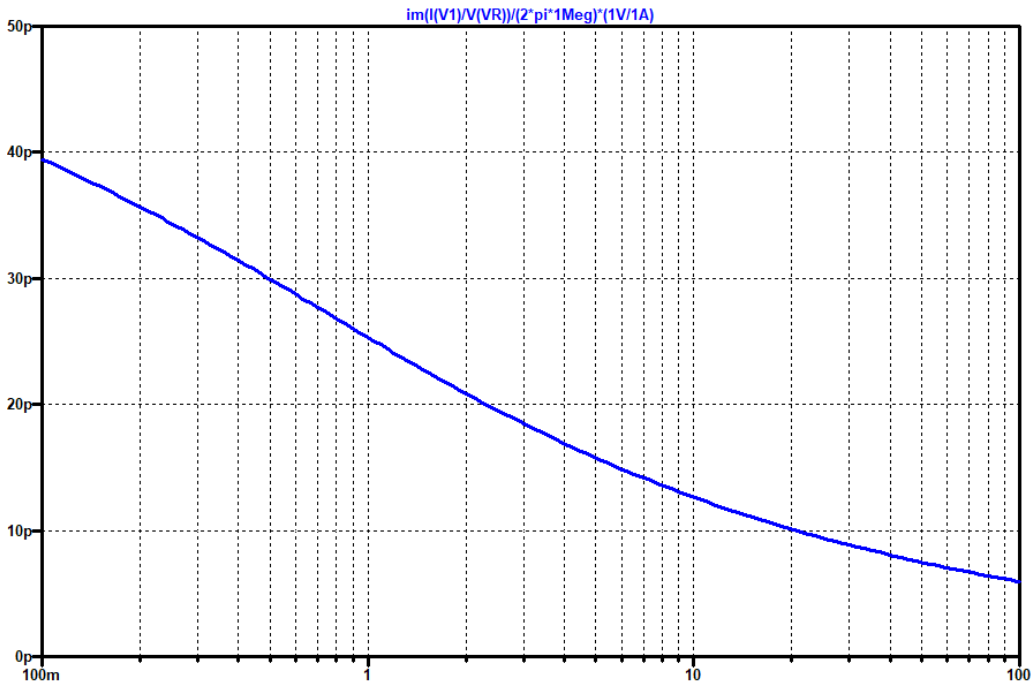
TestBench

```
.step dec param VR 0.1 100 10
.ac lin 1 1Meg 1Meg
.OPTION TNOM=25 GMIN=1e-15
.TEMP 25
```



Simulation results are following.

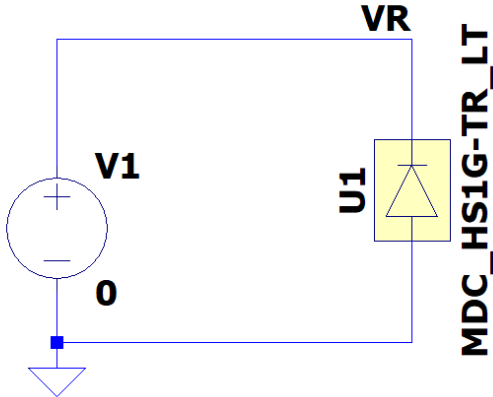
Explanatory notes — : simulated



Reverse Current – Reverse Voltage

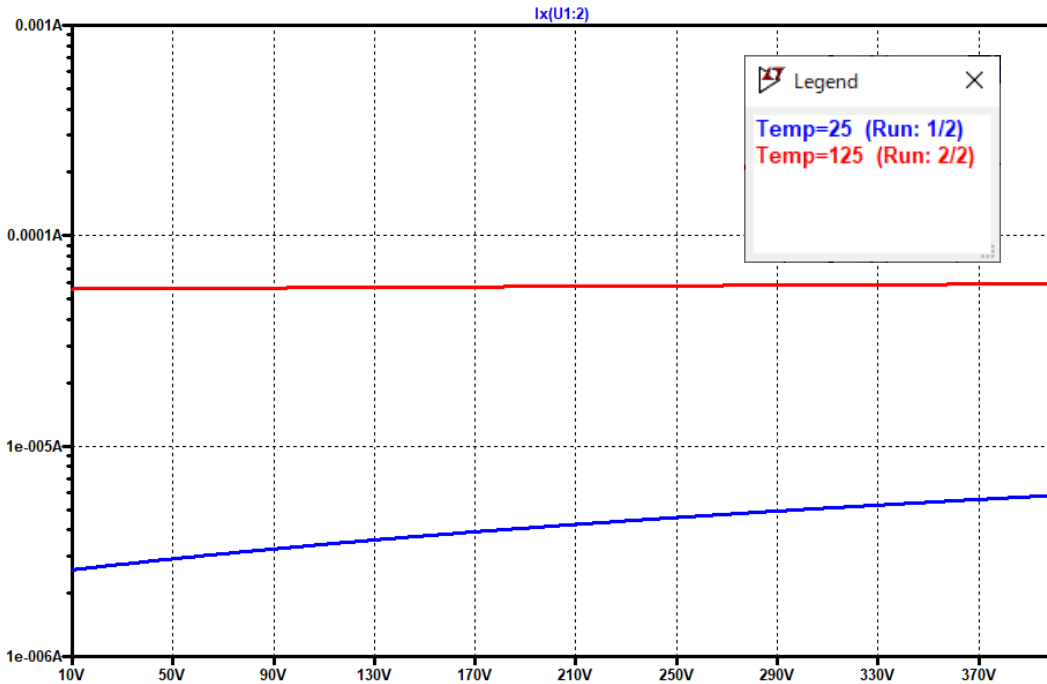
TestBench

```
.dc V1 0 400 1
.OPTION TNOM=25 GMIN=1e-15
.TEMP 25 125
```



Simulation results are following.

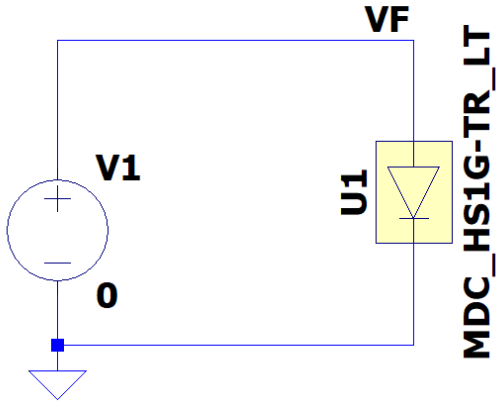
Explanatory notes — : simulated



Forward Current – Forward Voltage

TestBench

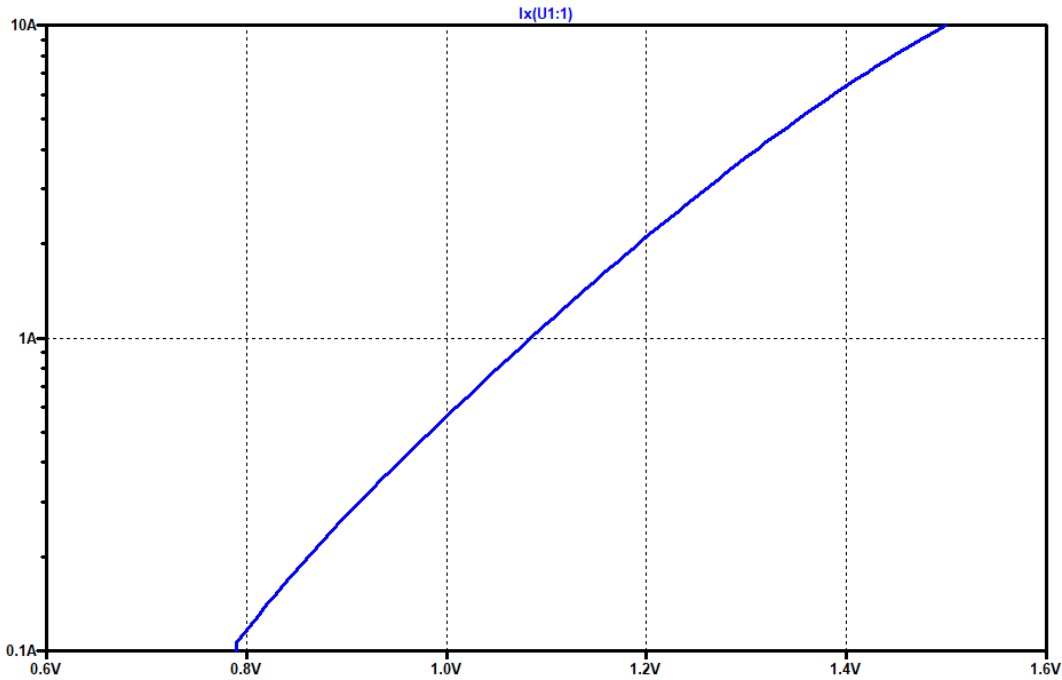
```
.dc V1 0.6 1.6 0.01
.OPTION TNOM=25 GMIN=1e-15
.TEMP 25
```



Reverse Recovery Time

Simulation results are following.

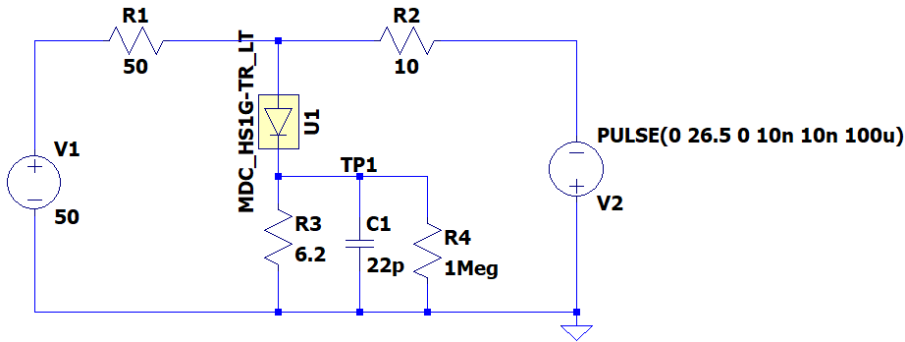
Explanatory notes — : simulated



Reverse Recovery Time

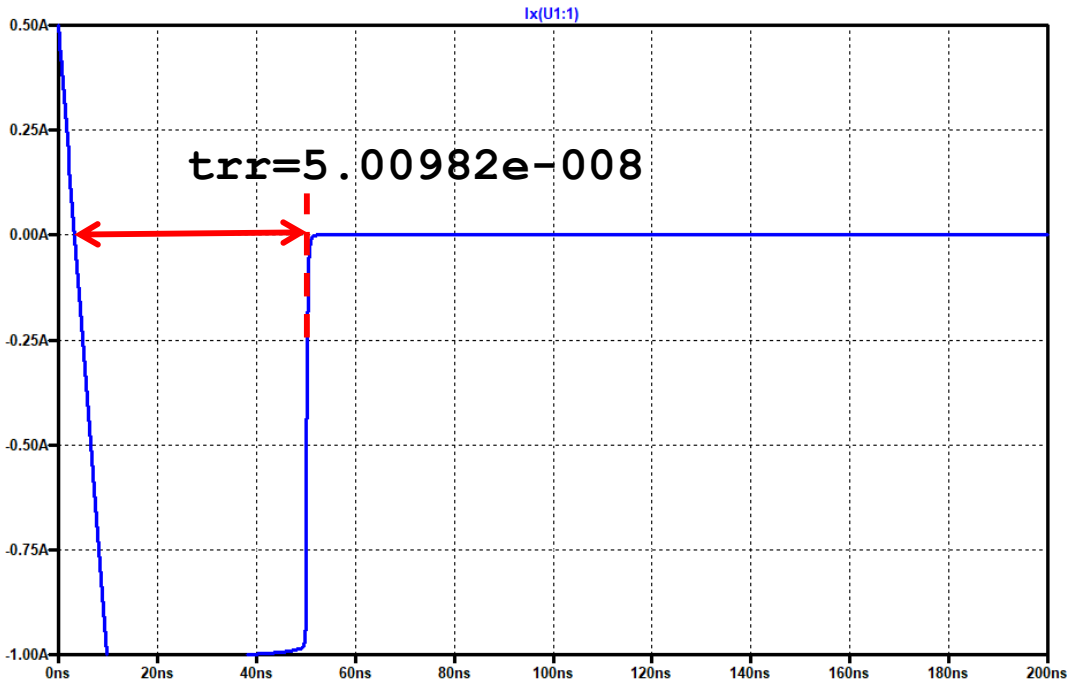
TestBench

```
.meas trr TRIG Ix(U1:1)=0 FALL=1 TARG Ix(U1:1)=-0.25 RISE=1
.tran 0 200n 0 0.1n
.OPTION TNOM=25
.TEMP 25
```



Simulation results are following.

Explanatory notes — : simulated



DISCLAIMER

1. This SPICE (Simulation Program with Integrated Circuit Emphasis) model and its content (the "Contents") are copyright of MoDeCH Inc. All rights reserved. Any redistribution or reproduction of any or all part of the Contents in any form is prohibited without express written permission made by MoDeCH Inc.
2. MoDeCH Inc. as licensor (the "Licensor") hereby grants to you, as licensee (the "Licensee"), a non-exclusive, non-transferable license to use the Contents as long as you abide by the terms and conditions of this DISCLAIMER.
3. The Licensee is not authorized to sell, loan, rent and redistribute or license the Contents in whole or in part, or in modified form, to anyone.
4. The Licensor shall in no way be liable to the Licensee or any third party for any loss or damage (including ,but not limited to, lost profits, or other incidental, consequential, or punitive damages), however caused (including through negligence) which may be directly or indirectly suffered from, arising out of, or in connection with, any use of the Contents .
5. Notwithstanding anything contained in this DISCLAIMER, in no event shall Licensor be liable for any claims, damages or loss which may arise from the modification, combination, operation or use of the Contents with the Licensee's computer programs.
6. The Licensor does not warrant that the Contents will function in any environment.
7. The Contents may be changed or updated without notice. MoDeCH Inc. may also make improvements and/or changes in the products, pricing and/or the programs related to the Contents at any time without notice.



MoDeCH Inc.

Head Office

Location: 5-15 Yokoyama-cho, Hachioji-Shi, Tokyo 192-0081, Japan

Tel:+81-42-656-3360

E-Mail:model-on-support@modech.co.jp

URL:<http://www.modech.com/en/>