

## LTspice Model Comparator STMicroelectronics TS391ILT

## **Model Information**

Model A macro model

Call Name MDC TS391ILT LT

Pin Assign 1:OUT 2:VCC- 3:IN- 4:IN+ 5:VCC+
File List Model Library MDC\_TS391ILT\_LT.lib

Model Report MDC\_. TS391ILT\_LTpdf(this file)

Verified Simulator Version LTspice XVII

Note

#### References

The information which was used for modeling is as follow:

[Data Sheet]

●Date/Version DS1551 - Rev 10 - January 2021

Product name TS391

Company nameSTMicroelectronics

[Characteristics listed]

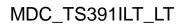
Characteristics

Input Offset Voltage Input Offset Current Supply Current Input Current Propagation delay

#### **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	deg C





O: Implemented

×: Not Implemented
—: Not applicable

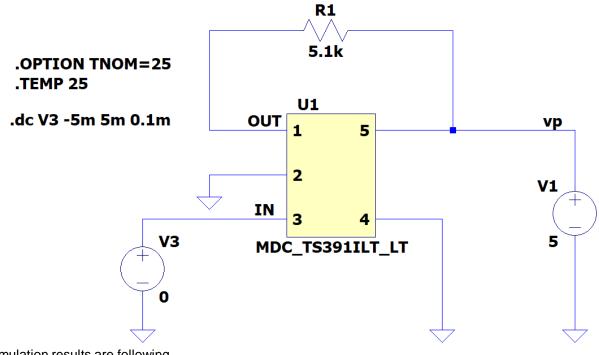
# Model Functions Table RANK=1

Functions	RANK	Implemented
Open Loop Gain	1	-
Input Offset Voltage	1	0
Input Offset Current	1	0
Bias Current	1	0
Maximum output amplitude voltage	1	_
Propagation delay	1	0



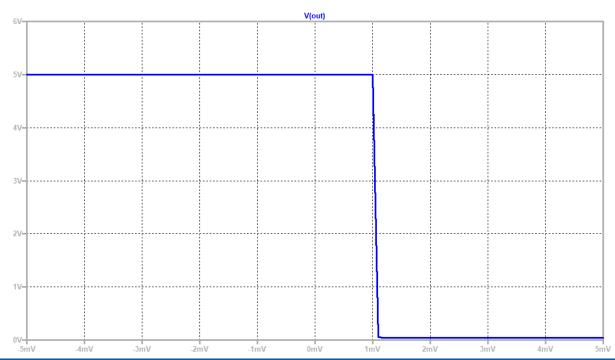
## **Input Offset Voltage Testbench**

## **Referred to Data Sheet**



Simulation results are following. Explanatory notes — : simulated

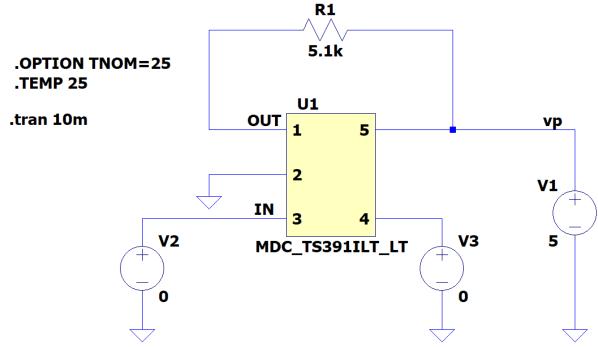
#### **Input Offset Voltage**





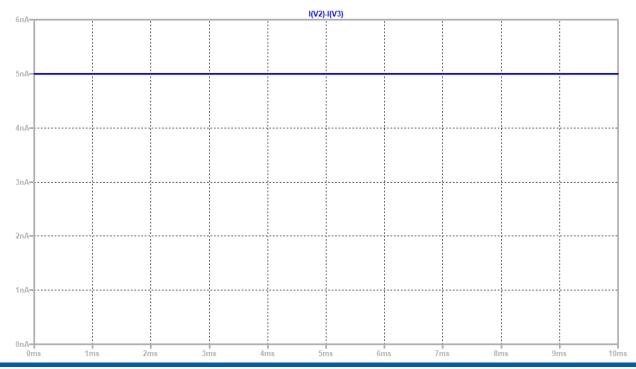
## **Input Offset Current Testbench**

## **Referred to Data Sheet**



Simulation results are following. Explanatory notes — : simulated

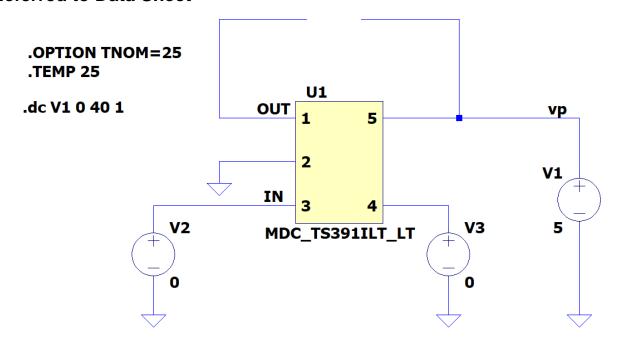
#### **Input Offset Current**





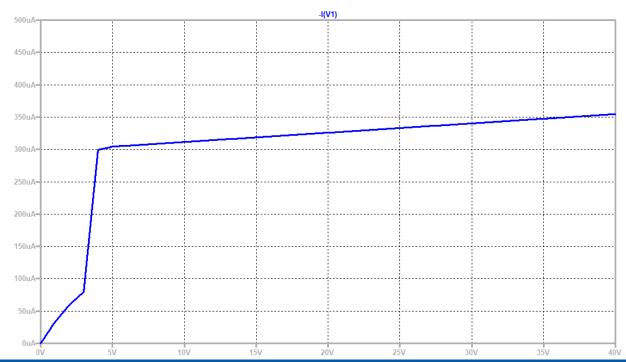
## **Supply Current Testbench**

## **Referred to Data Sheet**



Simulation results are following. Explanatory notes — : simulated

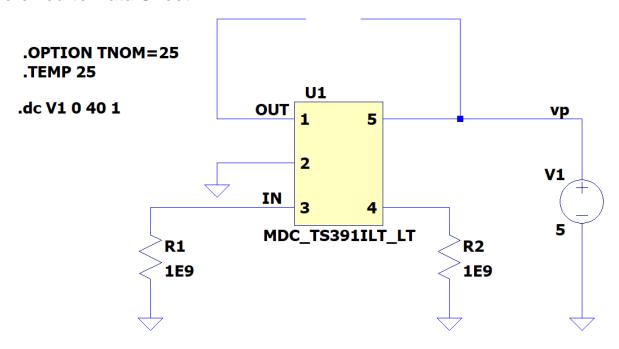
#### **Supply Current**





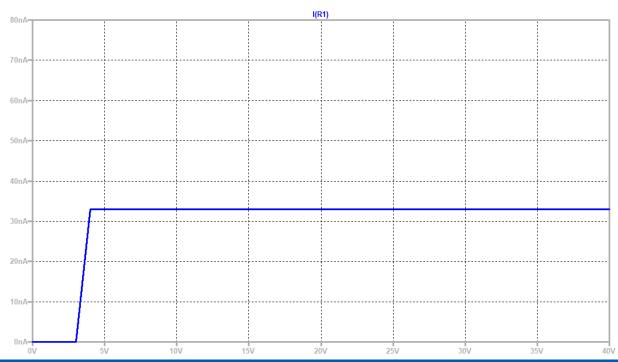
## **Input Current Testbench**

## **Referred to Data Sheet**



Simulation results are following. Explanatory notes — : simulated

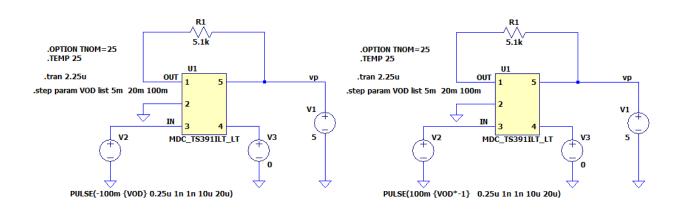
#### **Input Current**





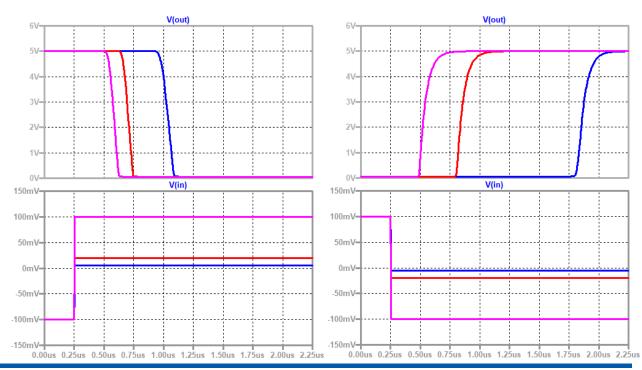
## **Propagation delay Testbench**

## **Referred to Data Sheet**



Simulation results are following. Explanatory notes — : simulated

#### **Propagation delay**





#### **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.
- MoDeCH Inc. as licensor (the "Licensor") hereby grants to you, as licensee (the "Licensee"), a nonexclusive, 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/