如何为隔离降压转换器选择变压器

Yaxian Li，高级应用工程师

摘要

本文阐述隔离式降压转换器的工作原理，以及应如何选择变压器，这是设计隔离式降压转换器的关键。本文探讨在选择变压器时应考虑哪些参数、应采用哪些数学公式进行运算，以及这些参数会如何影响整个电路。

隔离式降压转换器的工作原理

隔离式降压拓扑与通用降压转换器拓扑相似，如图1所示。使用变压器替换降压电路中的电感器，就可以得到隔离式降压转换器。变压器的副边可以独立接地。

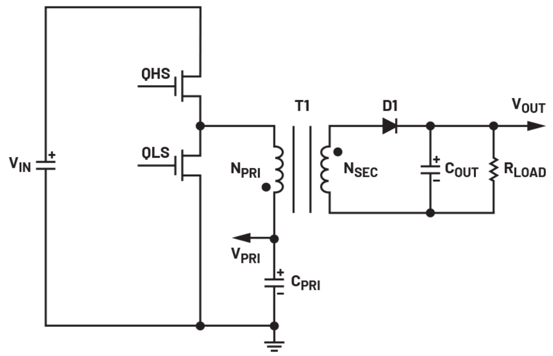


图1. 隔离式降压拓扑。

在导通期间，高侧开关管(QHS)接通，低侧开关管(QLS)断开。变压器的磁化电感(L_{PRI})此时被充电。电流方向如图2中的箭头所示。原边电流随时间线性增加。电流的斜率取决于(V_{IN} - V_{PRI})和L_{PRI}。在此时段内，副边二极管D1反向偏置，电流从输出电容(C_{OUT})传输至负载。

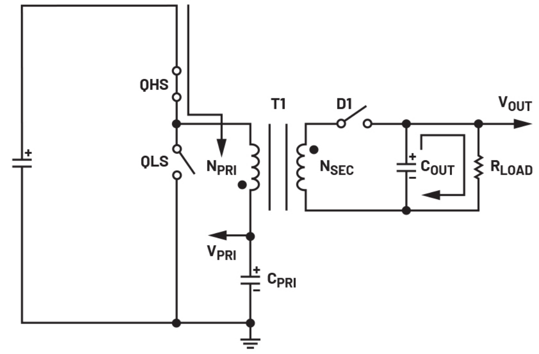


图2. 导通时段等效电路。

在关断期间，QHS断开，QLS接通。原边电感放电。原边电流从QLS流向地，D1正向偏置，副边电流从副边线圈流向C_{OUT}和负载。在此时段，输出电容（C_{OUT}）被充电。（断开QHS和接通QLS不会改变电流方向，只会改变电流坡度。在正电流降至0 A之后，反向电流开始增大。）

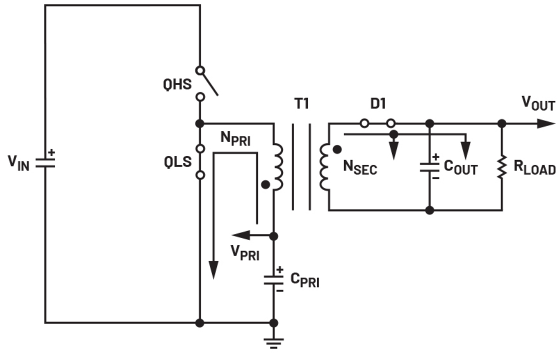


图3. 关断时段等效电路。

哪些规格参数将影响变压器选择?

在设计转换器时，应明确指定一些规格参数。它们将决定电路选用哪些器件尤其是选用那个变压器。

- 输入电压范围
- 输出电压
- 最大占空比
- 开关频率
- 输出电压纹波
- 输出电流
- 输出功率

最大占空比(D)通常介于0.4至0.6之间。最小输入电压(V_{IN_MIN})和最大占空比将决定原边输出电压(V_{PRI})。原边输出电压(V_{PRI})和副边输出电压(V_{OUT})将决定变压器的匝数比。

输出电流(I_{OUT})和输出功率(P_{OUT})是影响变压器选择的重要参数。输出电流决定铜线的粗细，输出功率决定应使用哪个变压器线轴。线轴的磁导率决定它能存储多少电能，能输出多少功率。一般来说，DC输出电流乘以一个系数，就能得出电感（变压器）的纹波电流。占空比和开关频率可以用来计算T_{ON}时间，V_{IN}、V_{PRI}和纹波电流可以确定原边电感。分配的系数不能太大或太小，较大的系数会导致高纹波电流。高纹波电流可能达到半H桥限流值的，这会损坏MOSFET。由于存在ESR和ESL，输出电容将会承受大纹波电压。与之相反，当需要极小的纹波电流时，应使用电感值很高的电感器（变压器）。如果线圈匝数很多，则需要使用笨重的线轴。大电感会限制环路带宽，降低动态响应系数。

选择变压器

电能仅在T_{OFF}时间内传输至副边线圈。匝数比可以通过公式1计算：

$$\frac{V_{OUT}+V_D}{V_{PRI}}=\frac{N_{SEC}}{N_{PRI}} \qquad (1)$$

其中，V_D表示副边二极管正向偏置电压。V_{PRI}的最大占空比一般介于0.4至0.6之间。V_{PRI}可以采用公式2计算：

$$V_{PRI}=D\times V_{IN\_MIN} \qquad (2)$$

其中，D为最大占空比，V_{IN_MIN}为最小输入电压。使用公式2可计算出匝数比。在非隔离降压转换器中，电感两侧的纹波电流相同。利用公式3可轻松计算出所需的电感。

$$L=\frac{(V_{IN\_MIN}-V_{OUT})\times D}{f\times\Delta I} \qquad (3)$$

其中f为开关频率，ΔI为纹波电流。如前所述，我们设定纹波电流 = DC输出电流 x 系数。

$$\Delta I=I_{OUT}\times K \qquad (4)$$

其中K为系数。但是，隔离式降压转换器拓扑中只有变压器，没有电感。如果其中的组件是变压器而非电感时，该怎么办？我们知道，电流比等于匝数比的倒数：

$$IPRI_{TOFF}=I_{SEC}\times\frac{N_{SEC}}{N_{PRI}} \qquad (5)$$

$IPRI_{TOFF}$是关断期间副边电流，它在关断期间会激励原边产生电流。应添加变压器双线圈电流，将其作为等效电感电流。

$$ILeq=I_{PRI}+I_{SEC}\times\frac{N_{SEC}}{N_{PRI}} \qquad (6)$$

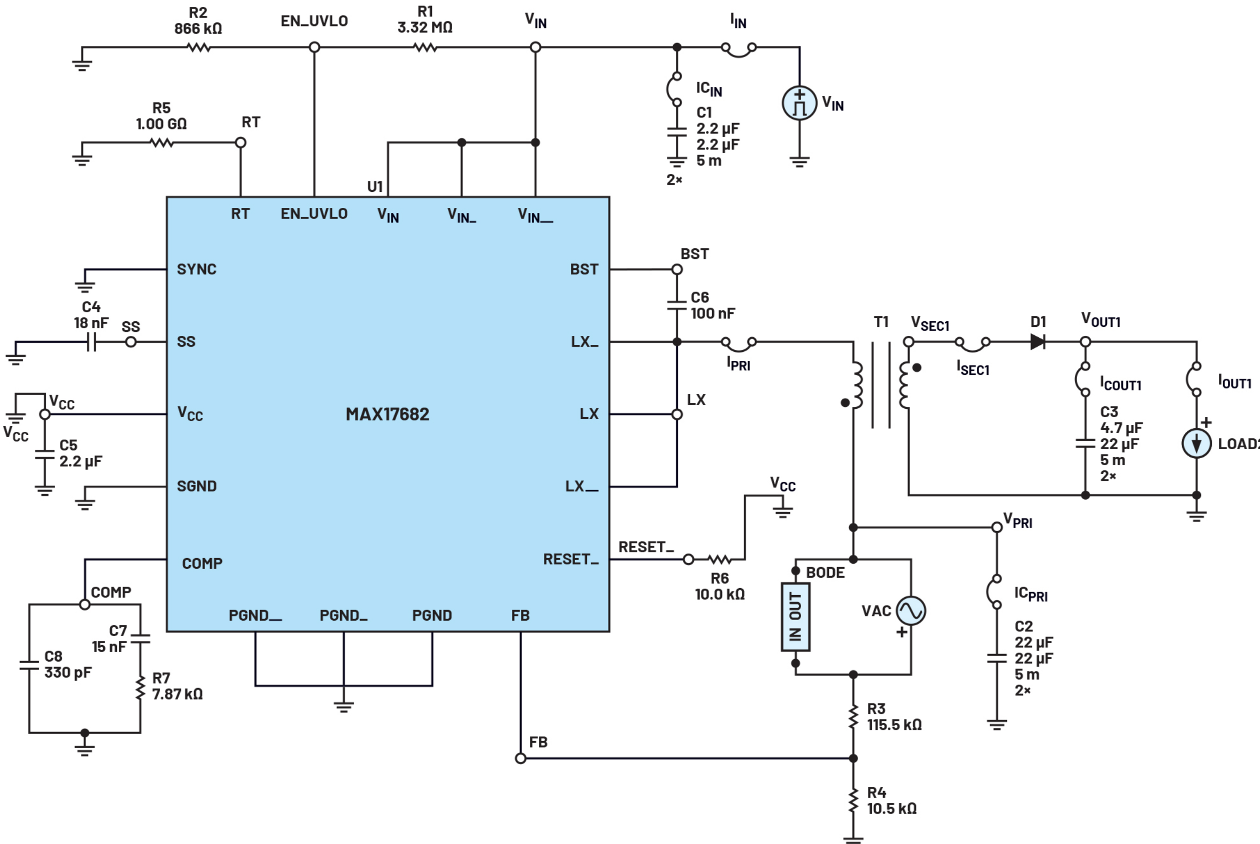
其中，ILeq表示等效电感电流。如果变压器还有三个绕组，那么

$$ILeq=I_{PRI}+I_{SEC}\times\frac{N_{SEC}}{N_{PRI}}+I_{THI}\times\frac{N_{THI}}{N_{PRI}}+\cdots \qquad (7)$$

是这样吗？我们来看看基于MAX17682得出的仿真结果。图4显示MAX17682电路，采用EE-Sim® OASIS绘制，由SIMetrix/SIMPLIS提供。变压器两侧装有电流检测探头，分别为I_{PRI}和$_{ISEC1}$。

See product information

Change design in online tool



MAX17682 Initial Conditions

Initial Conditions can speed up simulations, but are not always necessary for running most simulations.

Seven adjustable parameters, IC_RESET, IC_SSDONE, IC_EN, IC_COMP, IC_FB, IC_V_{REF}, and IC_CLK are built into the MAX17682 model for setting internal initial conditions. These parameters can be edited by double clicking the MAX17682, then entering the desired values in the GUI box that pops up. Click OK when done making edits.

For AC, POP (Steady State), Load Step and Line Transient Analyses, Initial conditions are as follows:
I_{OUT} is the load current
1. Initial condition of C2 is V_{PRI}.
2. Initial condition of C3 is V_{SEC}.
3. Initial condition of C4 is 5.
4. Initial condition of C5 is 5.
5. Initial condition of C6 is -5.
6. Initial condition of C7 is V_{COMP}.
7. Initial condition of C8 is V_{COMP}.
8. IC_RESET = IC_SSDONE = IC_EN = 5
9. IC_COMP = V_{COMP}
10. IC_FB = IC_V_{REF} = 0.9
11. IC_CLK = 1.78

For Start-up Analysis
1. Initial condition of C2, C3, C4, C5, C6, C7 and C8 is 0.
2. IC_RESET = IC_V_{REF} = IC_SSDONE = IC_EN = 0
3. IC_COMP = IC_FB = 0
4. IC_CLK = 0

$dI = V_{PRI} \times (1 - (V_{PRI}/V_{IN}))/(Lm \times Fsw \times 1000)$
$IL = I_{PRI} + (I_{SEC} \times K)$
$V_{COMP} = 0.875 + (0.25 \times IL) + (0.125 \times dI)$

图4. MAX17682典型电路，采用EE-Sim OASIS绘制，由SIMetrix/SIMPLIS供电。

MAX17682 Notes
Circuit parameters for various simulation types

When downloaded from the on-line EE-Sim design tool, this file is configured for the simulation type selected in the downloading processes. It can easily be modified for other simulation types.

If you are new to SIMPLIS or simulation in general and you want to run a different kind of simulation (Load step, Line Transient, AC, Steady State, Start Up) following how it was done on-line, you can always download a separate schematic for each available simulation type as separate files. Doing so has the advantage that all simulation options and parameters, source and load parameters as well as initial conditions are set up for that type of simulation.

You can also to go to the menu item Simulator > Choose Analysis and set up the simulation parameters appropriately for the desired simulation. The load and source parameters and initial conditions are set by editing the parts on the schematic. The following are instructions and details you will need for modifying the schematic & simulator settings to run different simulations yourself.

The BODE and VAC devices are in place to allow measuring the control loop with AC simulations. Their presence does not adversely affect the other simulations. In a real circuit they would be replaced with a short.

LOAD1 and LOAD2 have several parameters to modify for different simulations. The load acts as a resistor in parallel with a pulsed current for transient Load Step simulations. When a Load Step/Pulse is not used the load acts as a resistor.

Double-click the load for a pop-up window with editable parameters for timing and amplitude of the pulse.
1. - 4.) Delay Time, Rise Time, Pulsewidth and Fall Time are used to set the timing of the pulsed load
5. Source Resistance sets the lower current of the pulse waveform; Current = V_{OUT}/Source Resistance
6. Start Current: This parameter informs the simulator what the lower current is per the prior step
7. Pulsed Current determines the higher peak current of the pulsed waveform.

The current set by the Source Resistance is present throughout the simulation. The Load device calculates the magnitude of the added pulse as (Pulsed Current - Start Current). In order to ensure the maximum current is the same as the value entered for Pulsed Current make sure the value entered for Start Current matches the current created by the Source Resistance; = V_{OUT}/Rsource.

There are two more Load parameters that are used but which are not included in the load pop-up GUI. These parameters can be edited by selecting the load, then right clicking on it and choosing "Edit/Add Propertiesâ€¦" from the pop-up list. When the Edit Properties windows opens, double click on the property you want to change. Change the value in the Edit Property window and click on OK. Then click on OK in the Edit Properties window.
1. ANALYSIS is a parameter that determines if the load pulse is used or if the load is a resistance
   1.1. For Load Step (Pulse) simulations set ANALYSIS to TRAN and the pulsed current will be used
   1.2. For all AC simulations and any transient simulations other than Load Step, set ANALYSIS to AC
2. AC_RSRC sets the load resistance value used when ANALYSIS = AC (TR_RSRC does so for TRAN)

The voltage source, V_{IN}, device is set to a DC voltage for all simulations other than Line Transient. When double clicked a GUI pops up with parameters that you can change. To make this into a DC source, set the Start Voltage and the Pulse Voltage to the same value

POP Simulation Settings. Accessible from the Simulator menu, Choose Analysis.

Needed for AC and Steady-state simulations as run online. Usually also used for Load Step and Line Transient Simulations. POP can be problematic at light load. If this is a problem for Load Step simulations you can set the starting current to be the higher level and the Step/pulse current to be the lower level.

Trigger Gate: X$U1.X$DRIVER.X$UPOP.!DCOMP
Max. Period: 10 µ
Cycles before launching POP: 100
Click the Advanced button for the next three parameters:
Convergence: 10 n
POP iteration limit: 20
Enable automatic transient analysis after a failed POP: Box is checked

V_{IN}, LOAD1, LOAD2, BODE & VAC are "test equipment" and not part of the circuit. BODE is replaced with a short in the real circuit.
Components representing open circuits: 1 GΩ Resistors, 1 fF Capacitors
Components representing short circuits: 1 mΩ Resistors

图4. MAX17682典型电路，采用EE-Sim OASIS绘制，由SIMetrix/SIMPLIS供电。

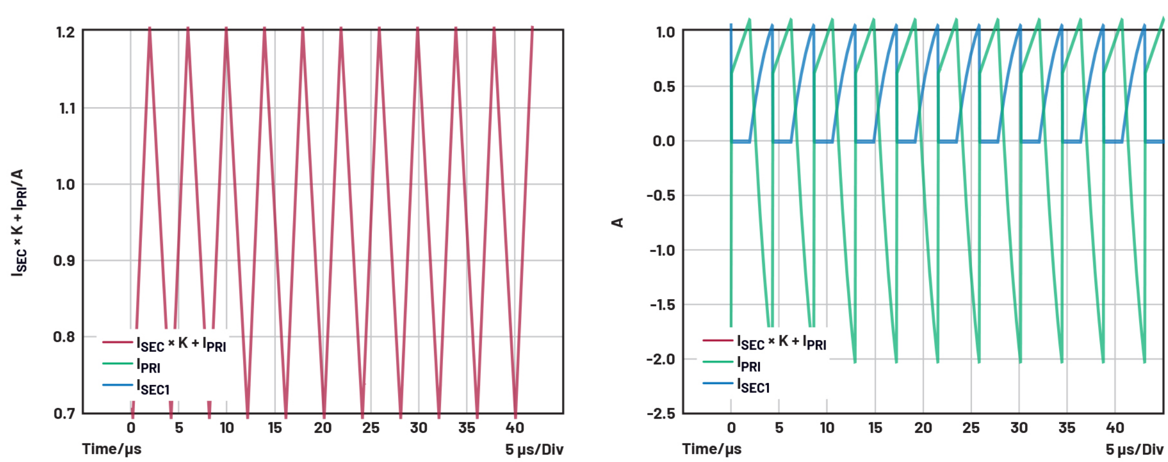


图5. MAX17682典型电路的仿真电流波形。

图5显示两个探头的瞬态仿真结果。这两个电流波形是根据公式6得出并添加的。

添加的电流结果（红色）为三角波，其表现就像是非隔离降压转换器中的电感。因此，可以使用以下公式轻松计算出变压器的原边ΔI：

$$\Delta I = \left(I_{PRI} + I_{SEC} \times \frac{N_{SEC}}{N_{PRI}}\right) \times K \tag{8}$$

通常，采用0.2倍DC输出电流作为负载纹波电流。所以，K等于0.2乘NSEC/NPRI。同时，原边峰值电流应低于开关限流值，其中I_{PK}的计算公式如下：

$$I_{PK} = ILeq_{DC} + \frac{\Delta I}{2} \tag{9}$$

变压器的原边电感可以使用以下公式计算得出：

$$L_{PRI} = \frac{(V_{IN\_MIN} - V_{PRI}) \times D}{f \times \Delta I} \tag{10}$$

根据匝数比、原边电感、输出功率、输出电流和隔离电压，可以决定应使用哪个电感。

为何简化公式是有效的

下面介绍如何更好地理解和使用MAX17682数据手册中所示的公式（参见图6）。

Primary Inductance Selection

Primary inductance value determines ripple current in the transformer. Calculate required primary inductance using the equation:

$$L_{PRI} = \frac{V_{PRI}}{f_{SW}}$$

where V_{PRI} and f_{SW} are nominal values.

图6. MAX17682数据手册的屏幕截图。

如前所述，可以将公式10改写为公式11，以得出关断期间的等效公式。

$$L_{PRI} = \frac{V_{PRI} \times (1 - D)}{f \times \Delta I} \tag{11}$$

假设D为0.6，只有当ΔI为0.4 A，多项式(1 - D)和ΔI才会抵消。公式11和图6所示的公式相同。数据手册中的公式已选择原边纹波电流。如果指定D为0.6，那么原边纹波电流为0.4 A。在数量上，T_{OFF}占空比等于原边纹波电流。

$$\Delta I = 1 - D \tag{12}$$

结论

根据图6所示的简化公式，指定原边纹波电流等于T_{OFF}占空比时，可以更快完成设计。如果想更改原边纹波电流或使用其他参数，可以按照本教程说明进行操作。

作者简介

Yaxian Li是ADI公司培训和技术服务团队的应用工程师。他专攻GMSL和RF技术。Yaxian于2020年加入Maxim Integrated（现为ADI公司一部分），于2018年获得杭州电子科技大学电气工程和自动化学士学位。他擅长打羽毛球和游泳。

TA24087sc-2/23(A)