

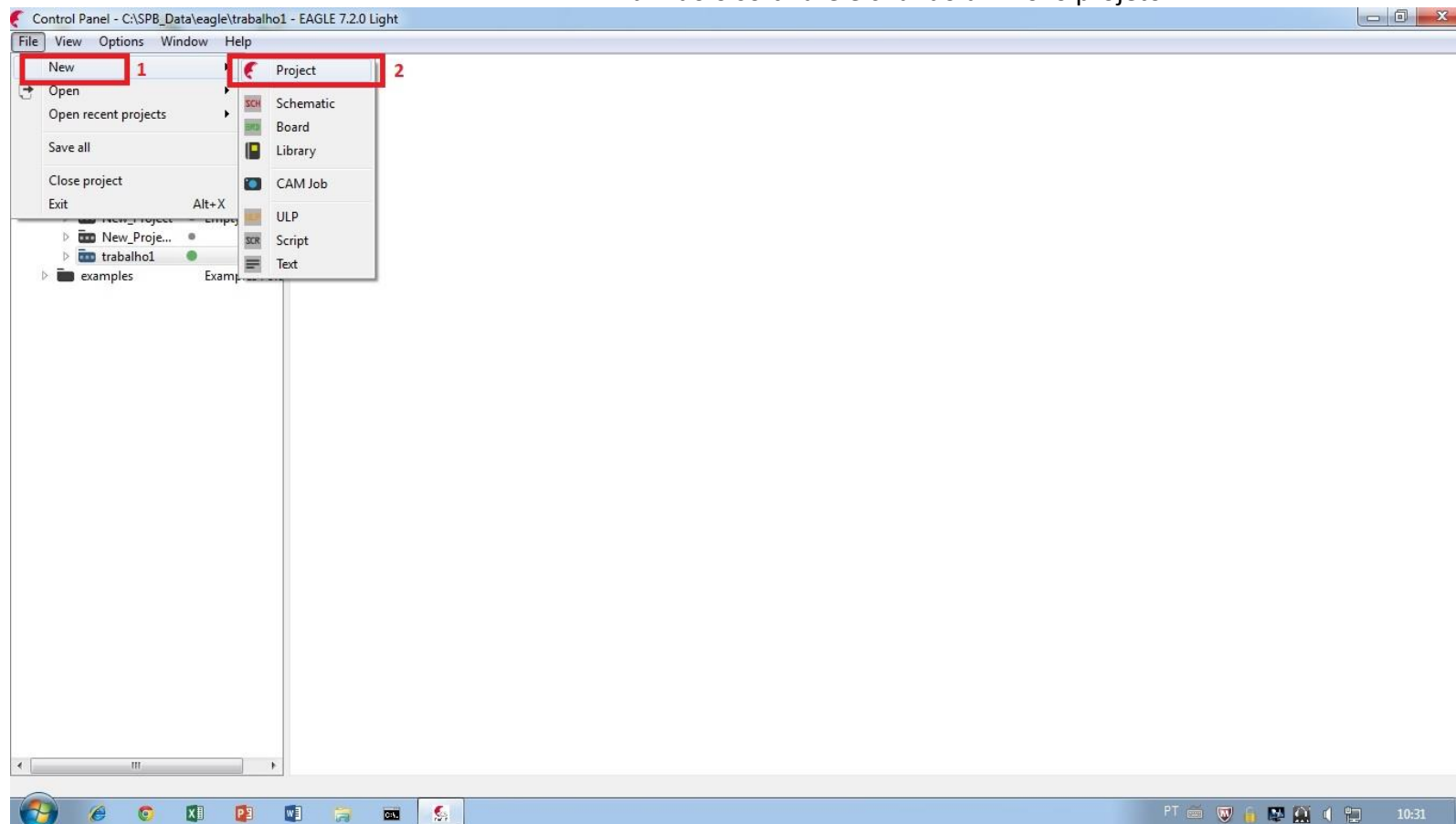


## TUTORIAL DO EAGLE 7.2 - 2015

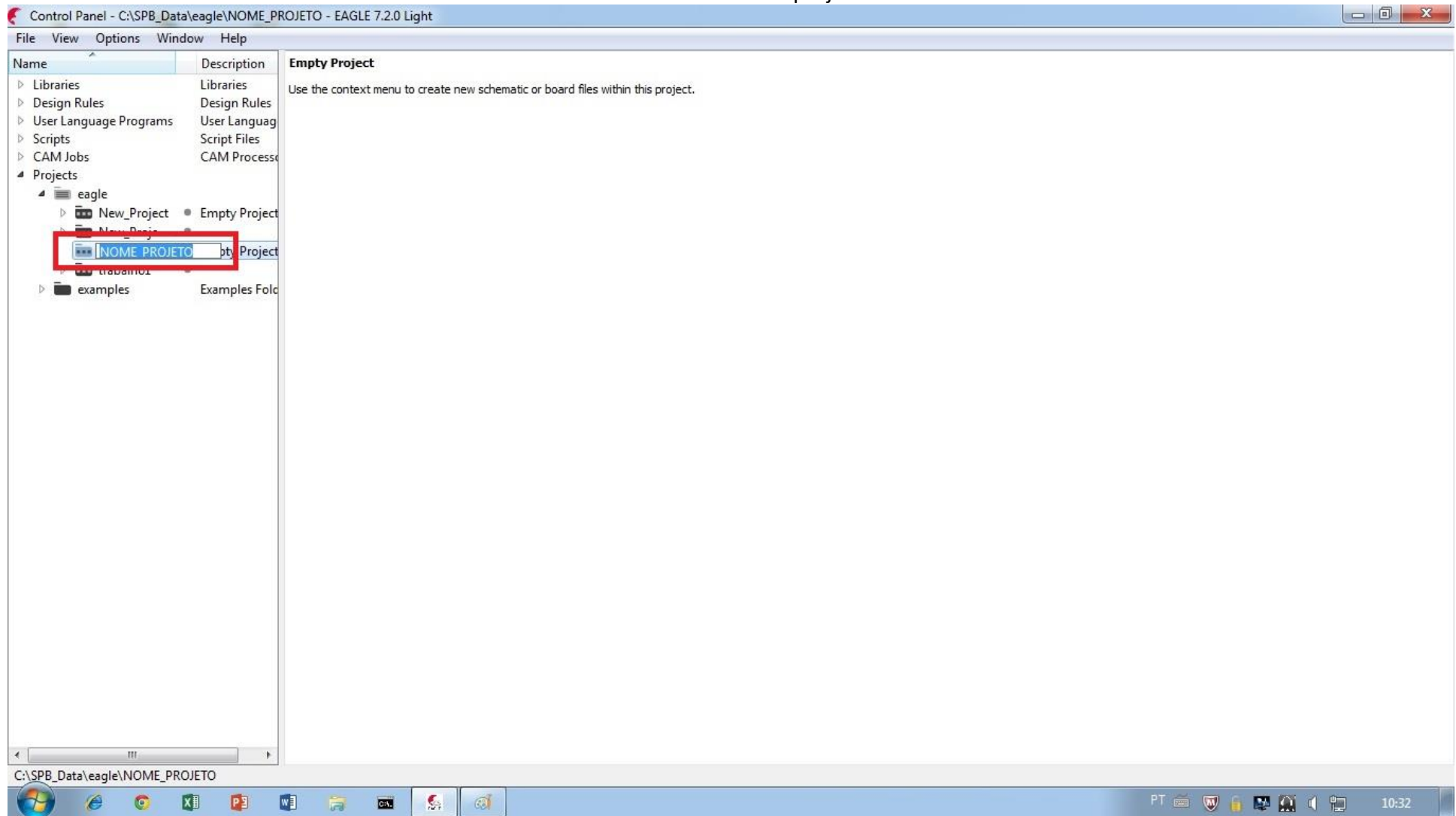
Prof. Juliano Benfica

Este tutorial visa a utilização do software de CAD EAGLE versão 7.2 que pode ser baixado do link <http://www.cadsoftusa.com/download-eagle/> para projeto de uma placa de circuito impresso face simples com esquemático e layout.

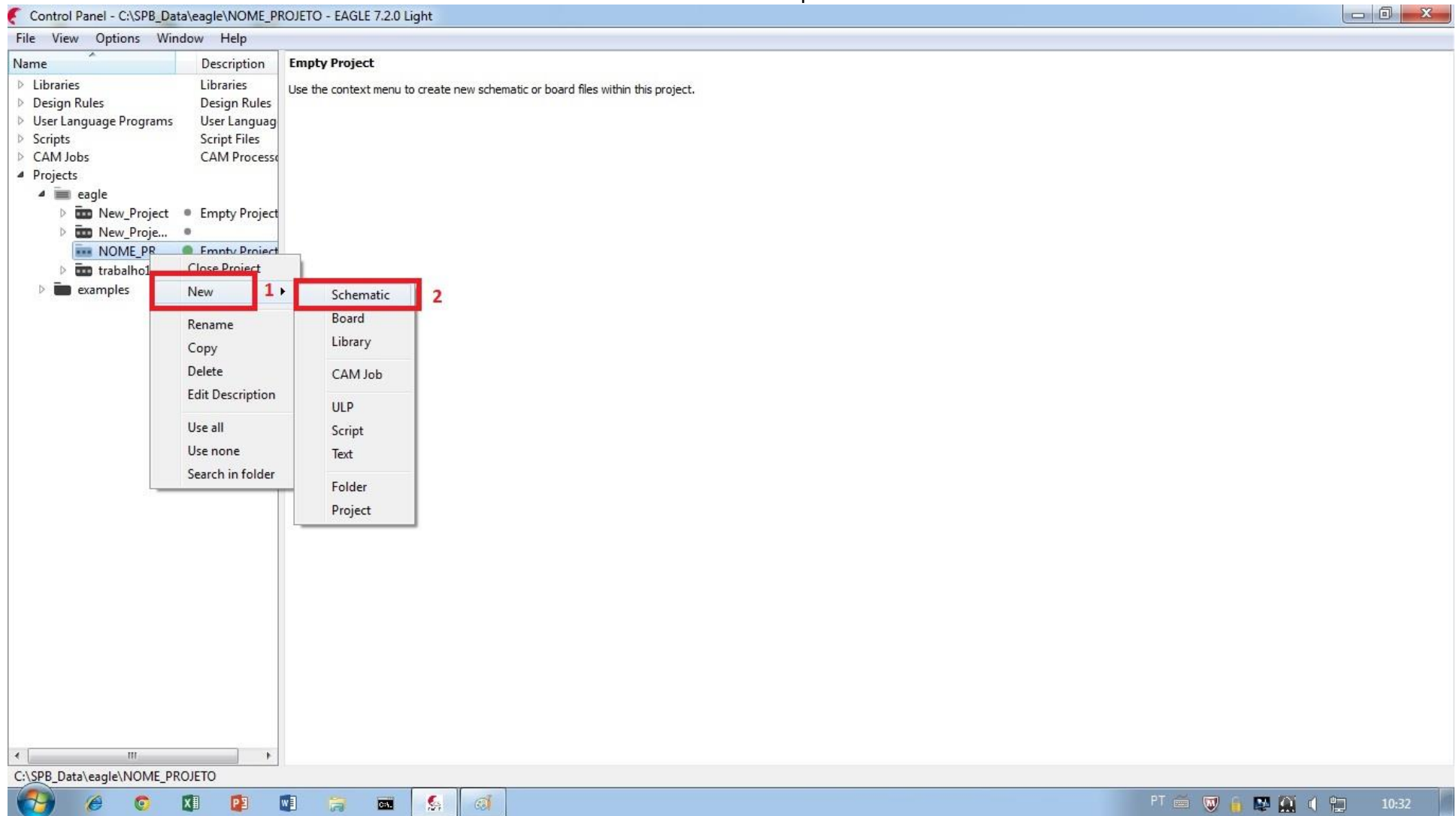
### 1 – Abrindo o software e criando um novo projeto



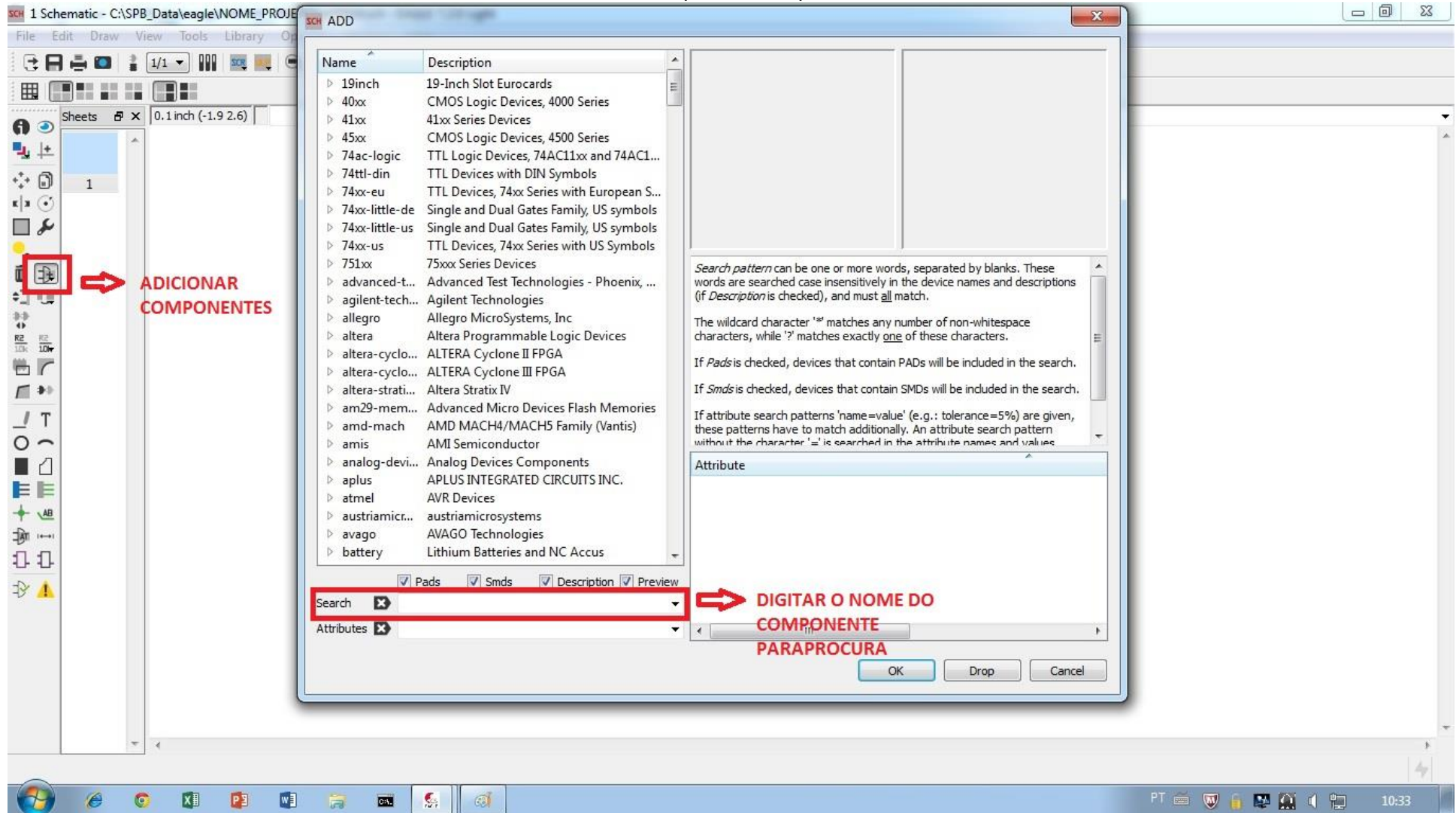
## 2 – Dando nome ao projeto



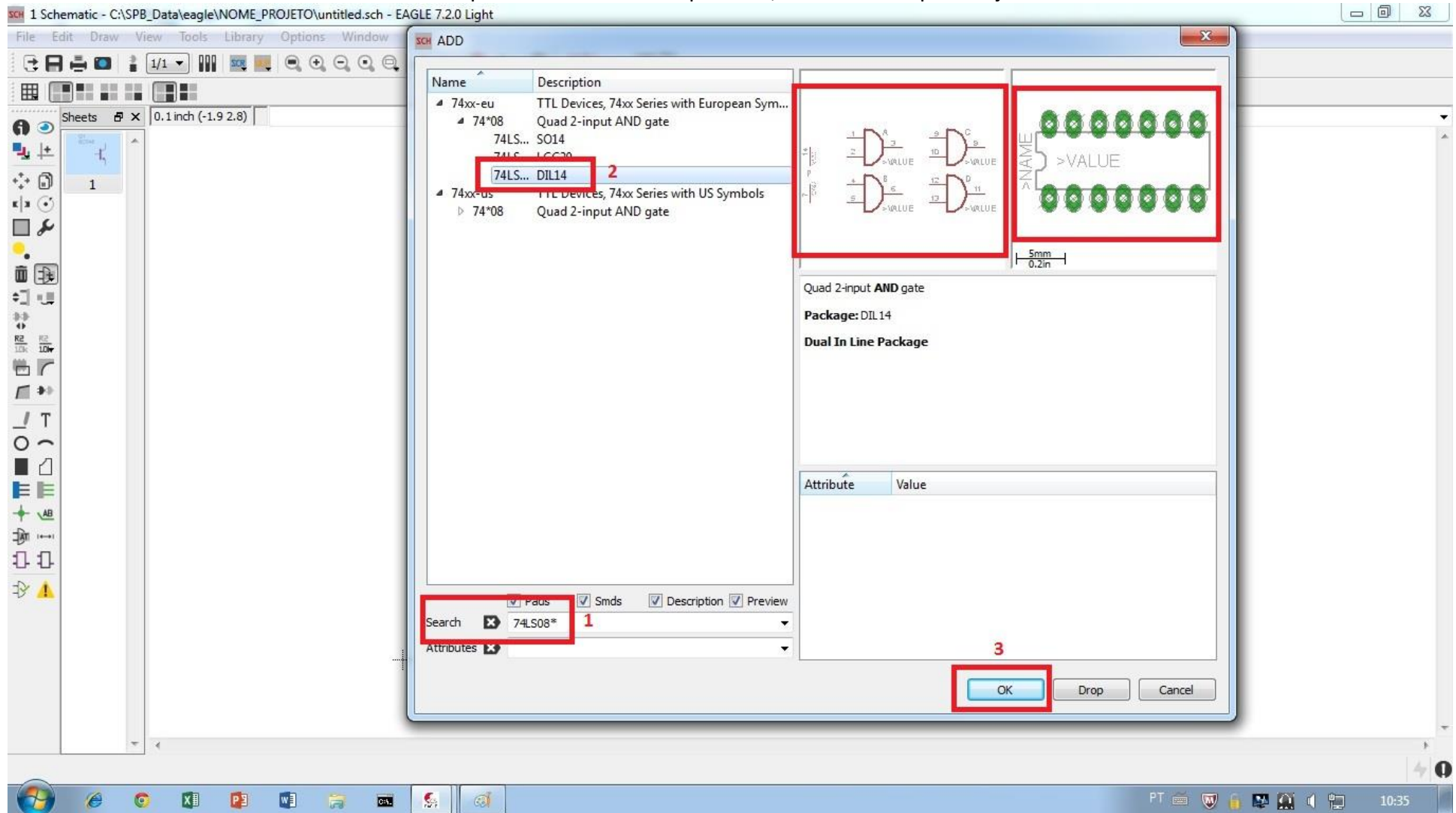
### 3 – Criando um novo esquemático



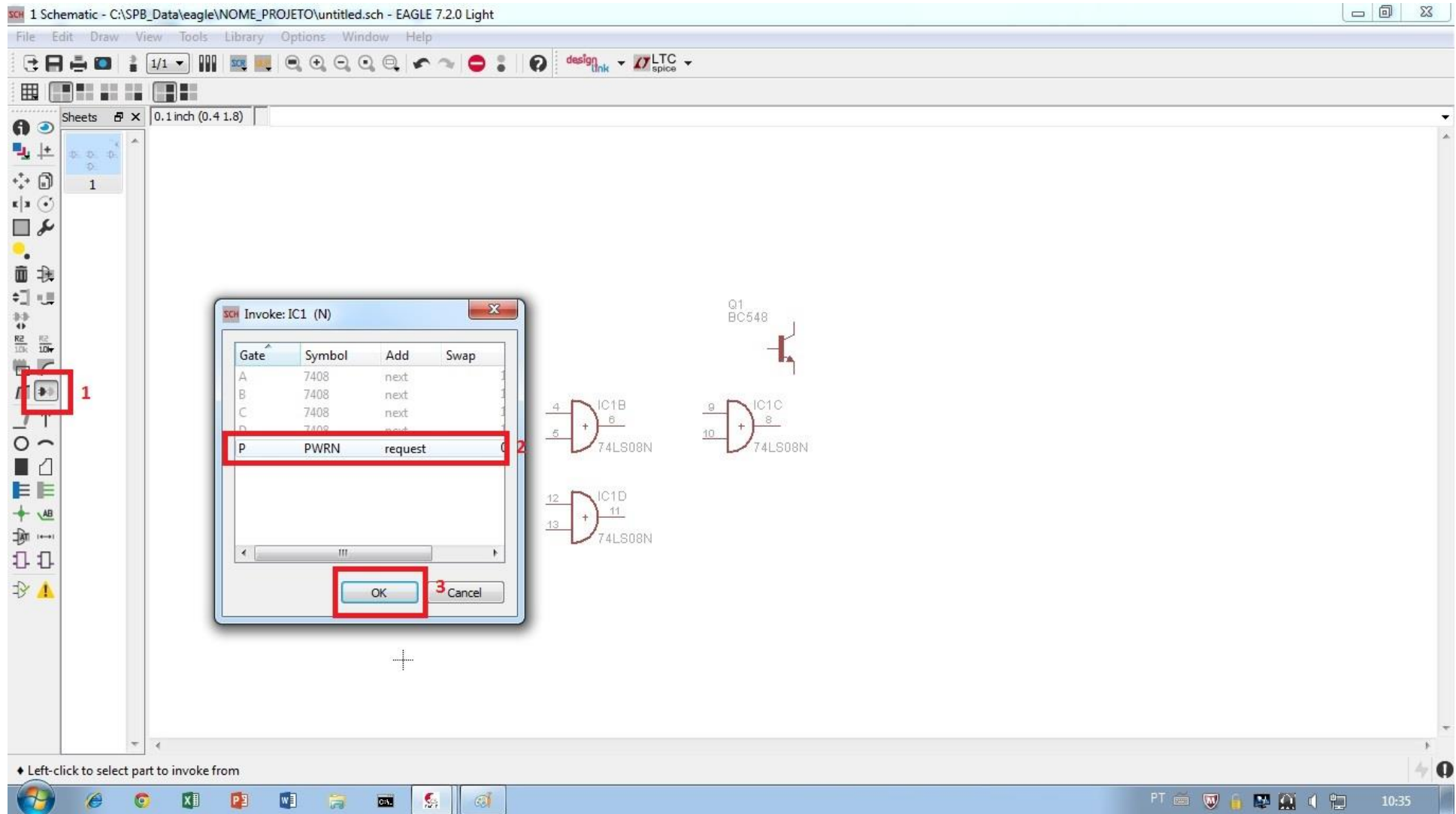
#### 4 – Adicionando componentes a partir da biblioteca



5 – Após encontrado o componente, selecionar o tipo desejado



## 6 – Desocultar os pinos de alimentação do CI.



## 7 – Seleccionando outro CI.

The image shows the Eagle 7.2.0 Light software interface. On the left, a schematic diagram shows a 74LS47N decoder connected to a +5V supply and ground. The decoder is labeled 'IC1' and has pins 7, 1, 2, 6, 4, 5, and 3 connected to the supply and ground. The output pins are labeled A through G. A component 'IC1G\$2' is also shown connected to the supply and ground.

The 'ADD' dialog box is open in the center, showing a list of components. The '74LS47N' component is selected and highlighted in blue. The dialog box also shows a preview of the component symbol and its physical package (DIL 16). The 'OK' button is highlighted in red.

The 'ADD' dialog box contains the following information:

Name	Description
74xx-eu	TTL Devices, 74xx Series with European Sym...
74LS47N	BCD to seven segment DECODER/DRIVER
74xx-us	TTL Devices, 74xx Series with US Symbols
74LS47N	BCD to seven segment DECODER/DRIVER

Search: 74ls47\* 1

Attributes: >NAME >VALUE

BCD to seven segment DECODER/DRIVER

Source: <http://www.ee.washington.edu/stores/DataSheets/74ls/74ls47.pdf>

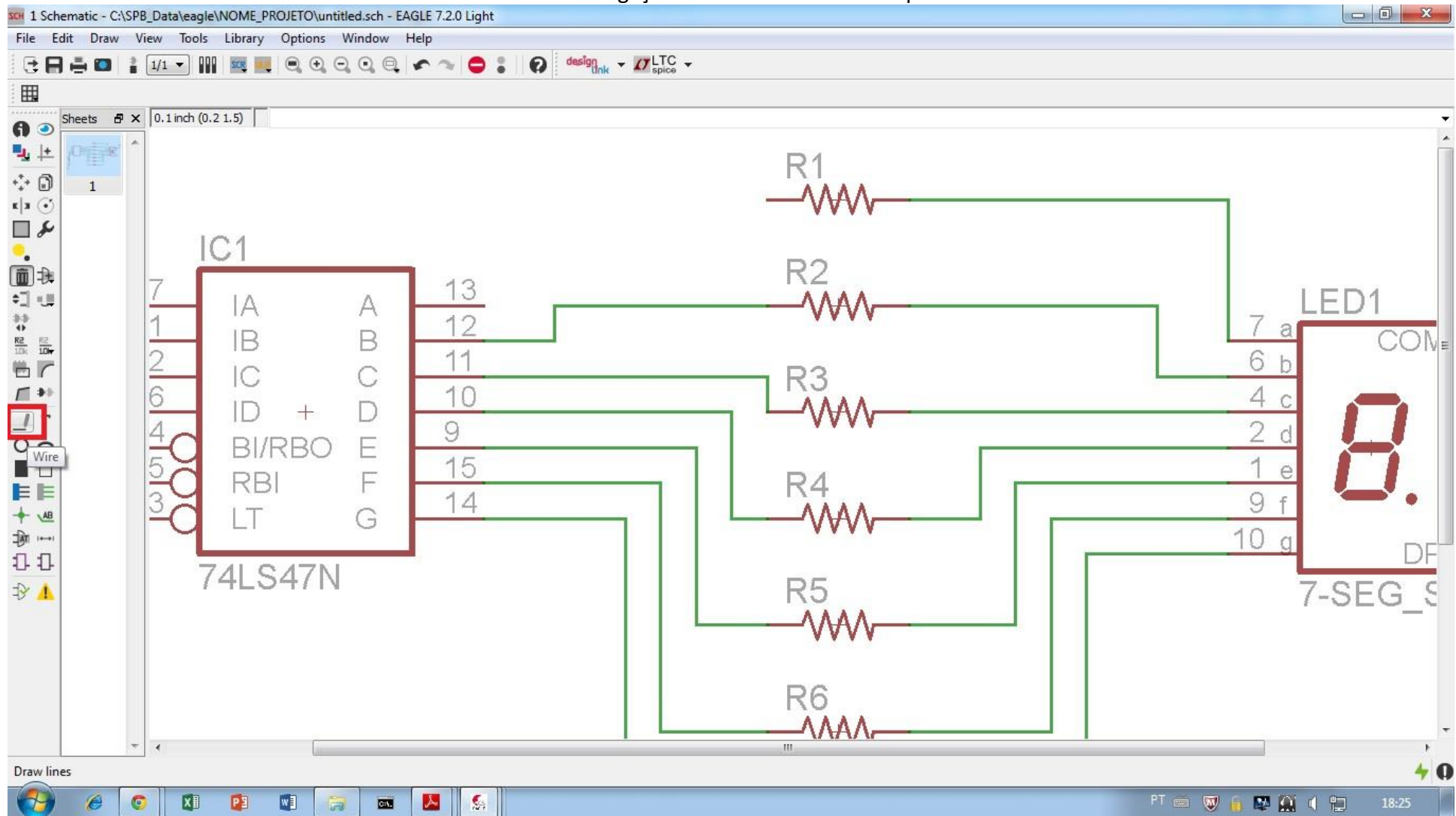
Package: DIL 16

Dual In Line Package

Attribute Value

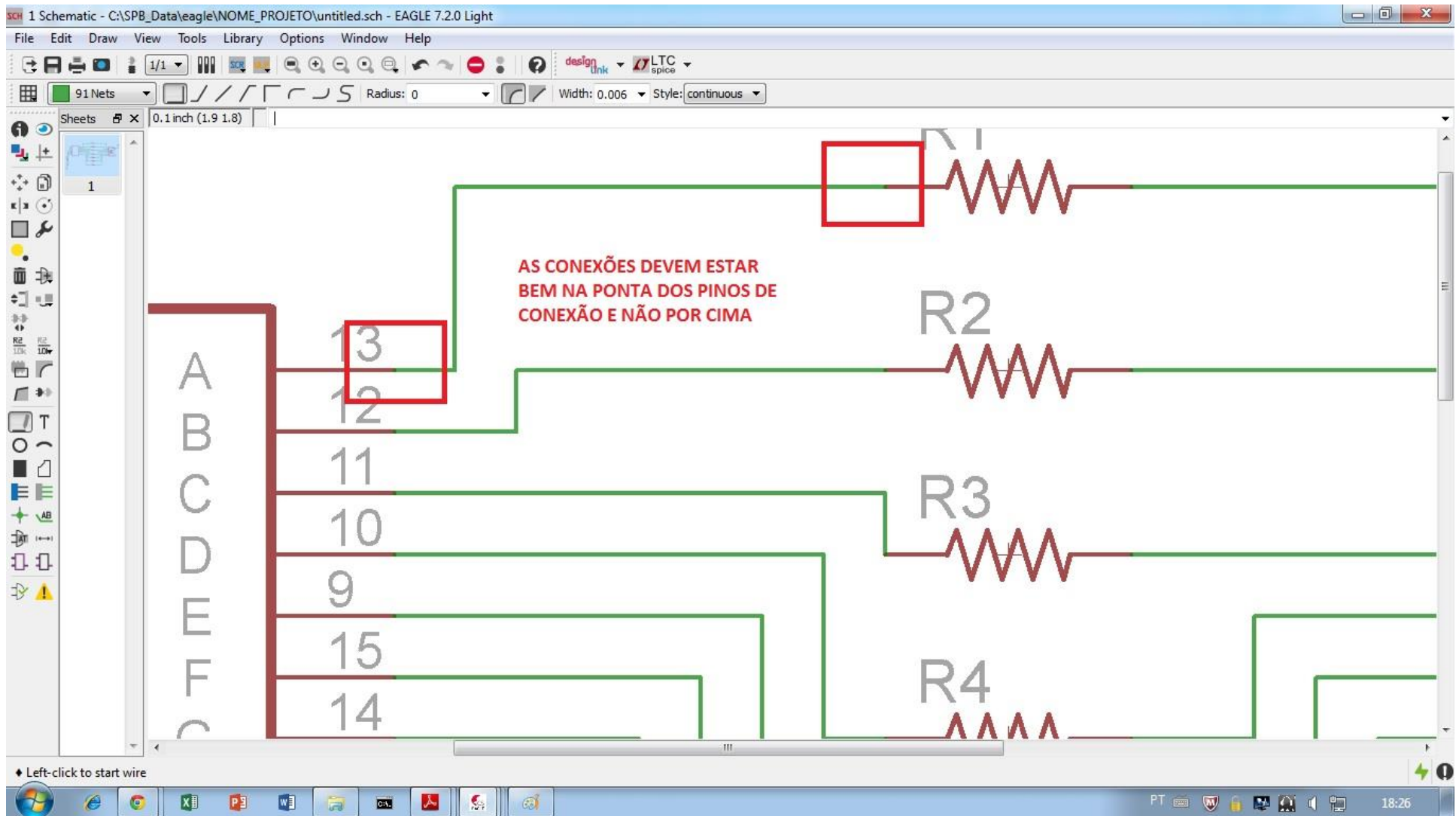
OK Drop Cancel

## 8 – Fazendo as ligações com fios entre os componentes.

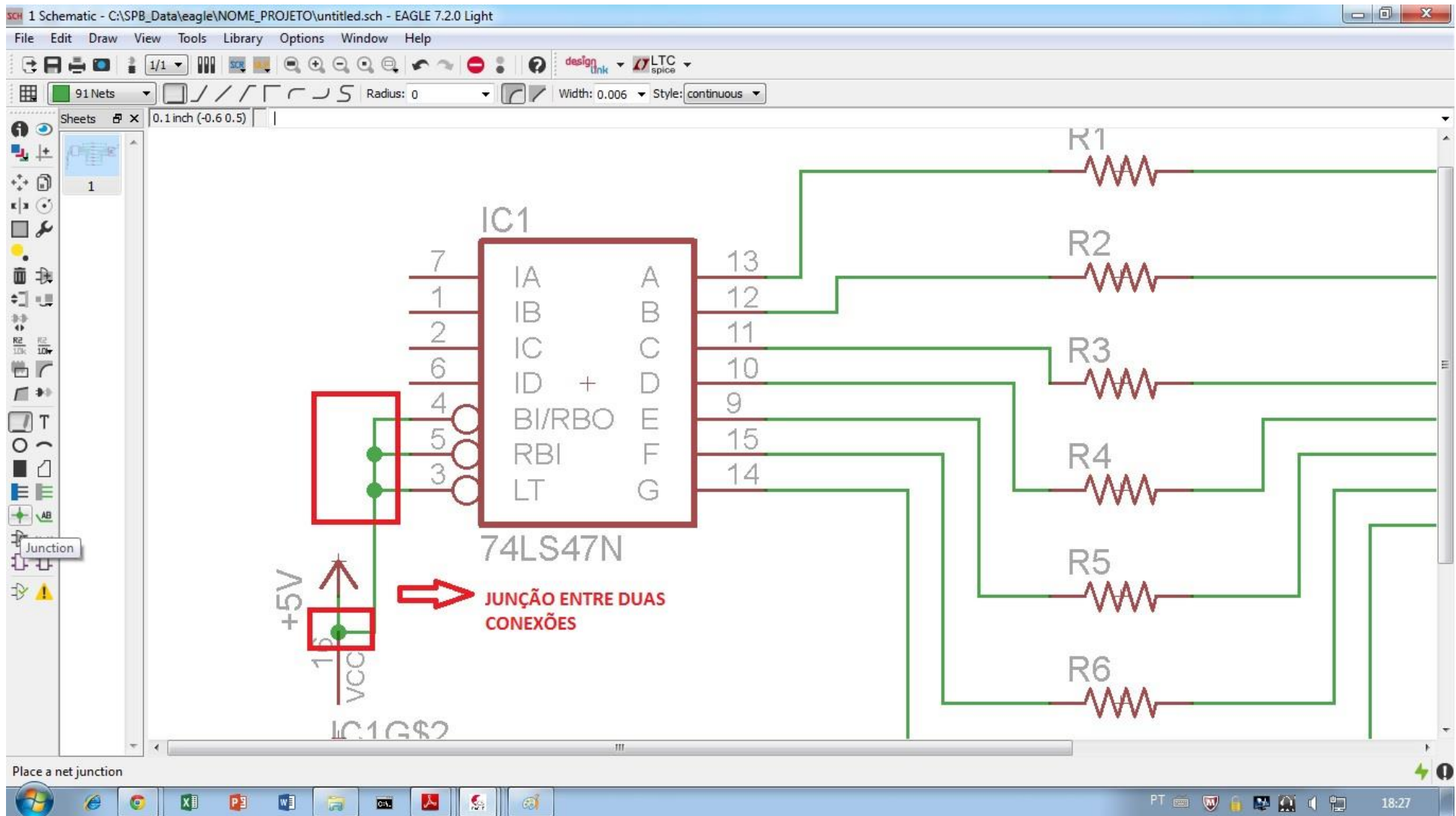




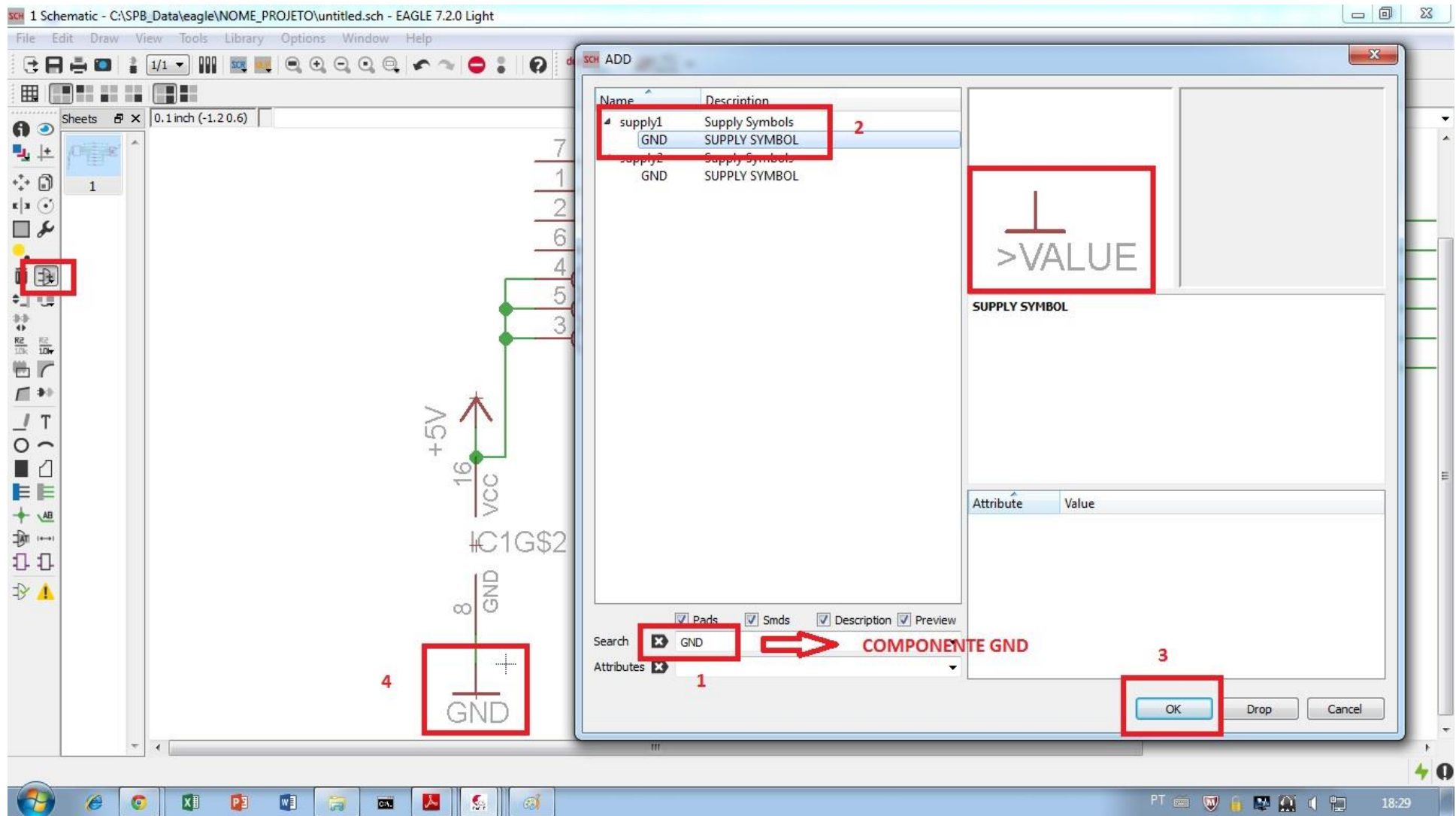
9 – Observar as conexões para que fiquem na ponta de cada componente.



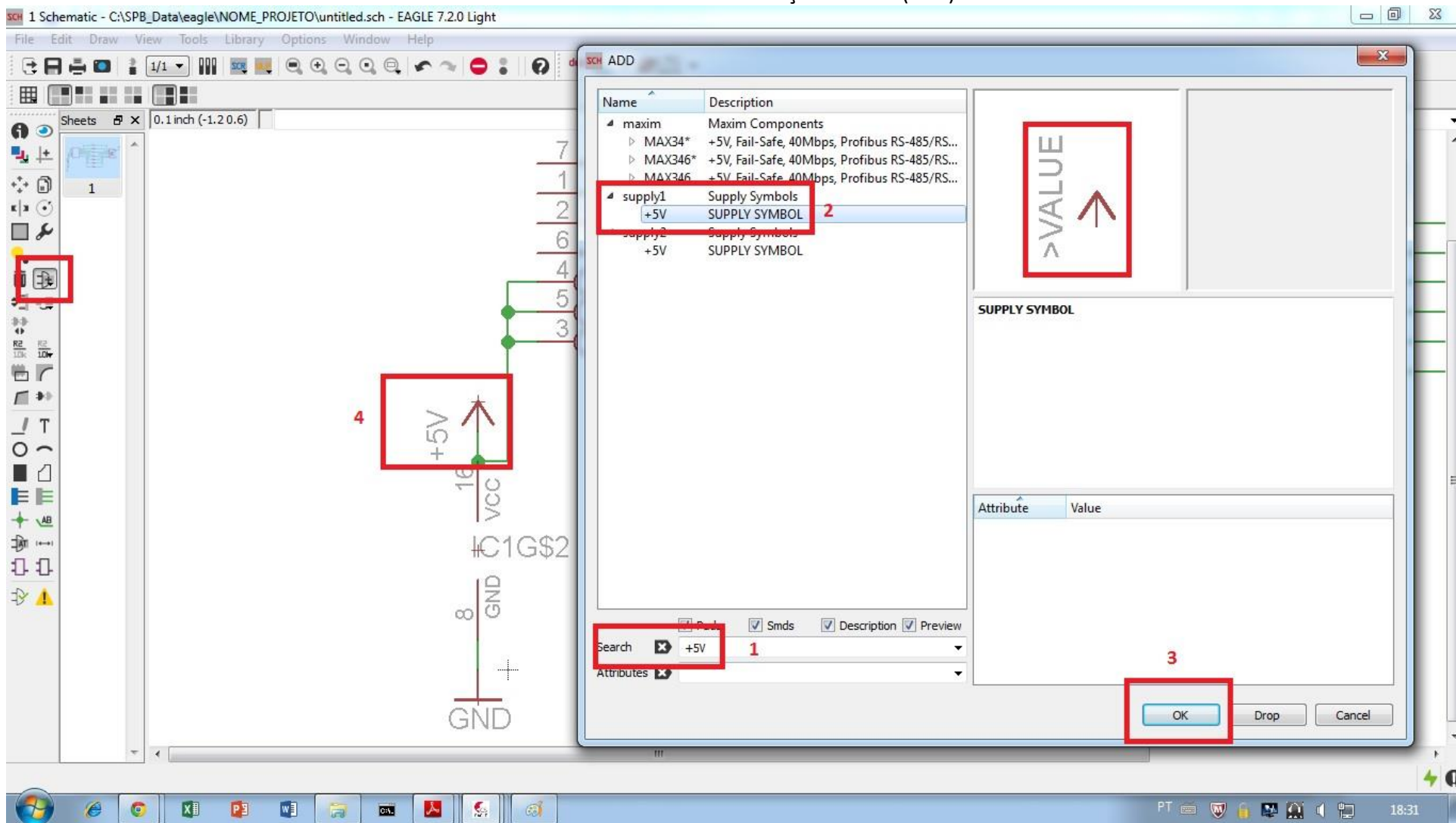
10 – Fazendo uma junção entre conexões, pois não é automático.



## 11 – Adicionando o terra (GND).



## 12 – Adicionando a alimentação de +5V (VCC).



### 13 – Deletando conexões e/ou componentes.

The screenshot shows the EAGLE 7.2.0 Light software interface. The main workspace displays a schematic diagram of a 74LS47N decoder (IC1) connected to a 7-seg LED display (LED1, 7-SEG\_SA56-11). The decoder's outputs (A-G) are connected to the display segments (a-g) through resistors R2 through R7. The decoder is powered by a +5V supply (VCC) at pin 16 and ground (GND) at pin 8. The display is also powered by a +5V supply at pin 3 and ground at pin 8. The decoder's inputs (IA, IB, IC, ID, BI/RBO, RBI, LT) are connected to pins 7, 1, 2, 6, 4, 5, and 3 respectively. The display's common pin (COM) is connected to +5V and the dot pin (DP) is connected to ground. A red box highlights the delete icon in the toolbar, with an arrow pointing to the text "PARA DELETAR TRILHAS E COMPONENTES".

**PARA DELETAR TRILHAS E COMPONENTES**

IC1  
74LS47N

LED1  
7-SEG\_SA56-11

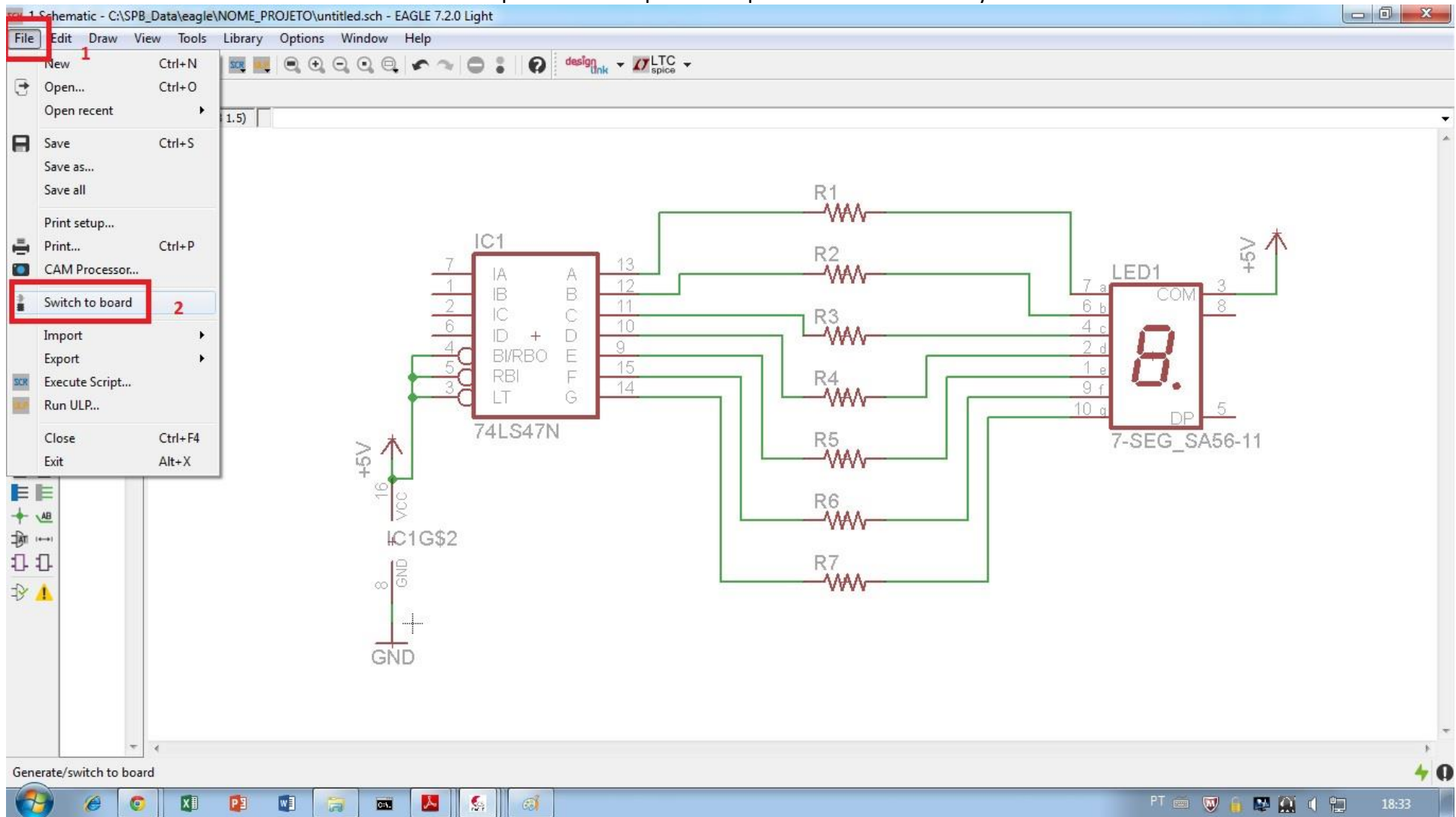
R2, R3, R4, R5, R6, R7

+5V, GND, VCC, GND

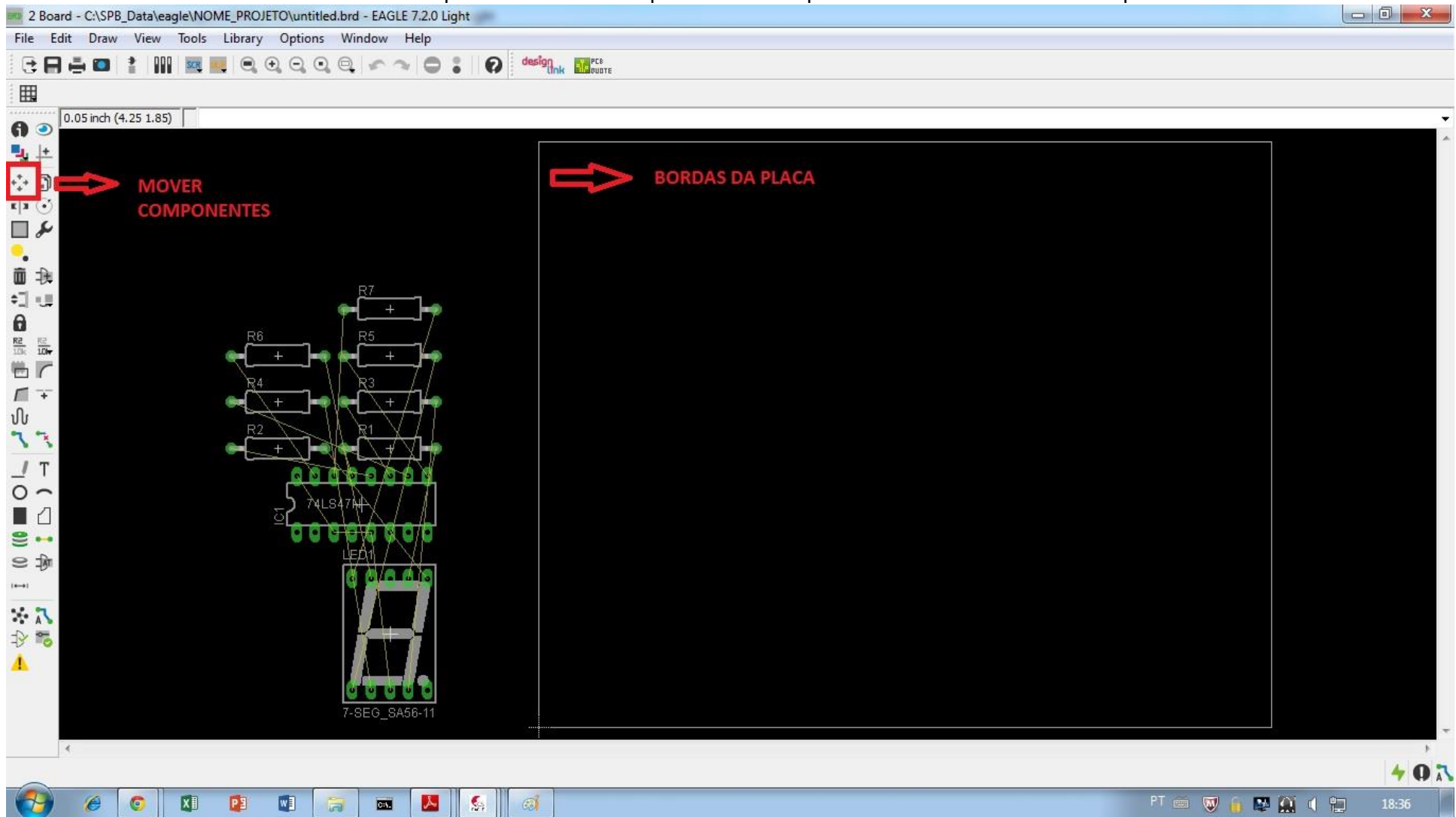
16 VCC, 8 GND, IC1G\$2

7 IA, 1 IB, 2 IC, 6 ID, 4 BI/RBO, 5 RBI, 3 LT, 13 A, 12 B, 11 C, 10 D, 9 E, 15 F, 14 G, 7 a, 6 b, 4 c, 2 d, 1 e, 9 f, 10 g, 3 COM, 8, 5 DP

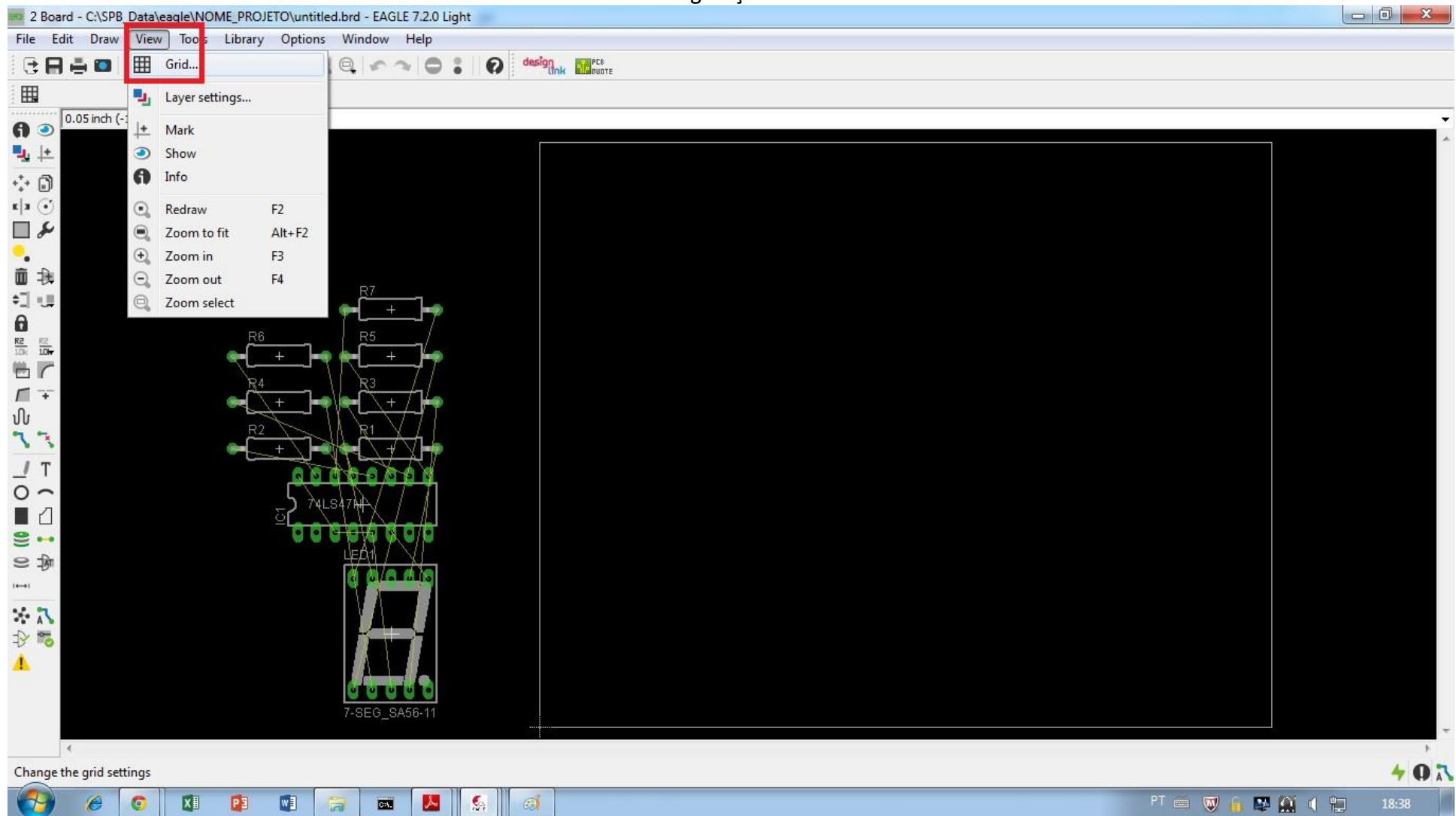
## 15 – Exportando o esquemático para a ferramenta de layout.



## 16 – Movendo e posicionando os componentes do esquemático dentro das bordas da placa.

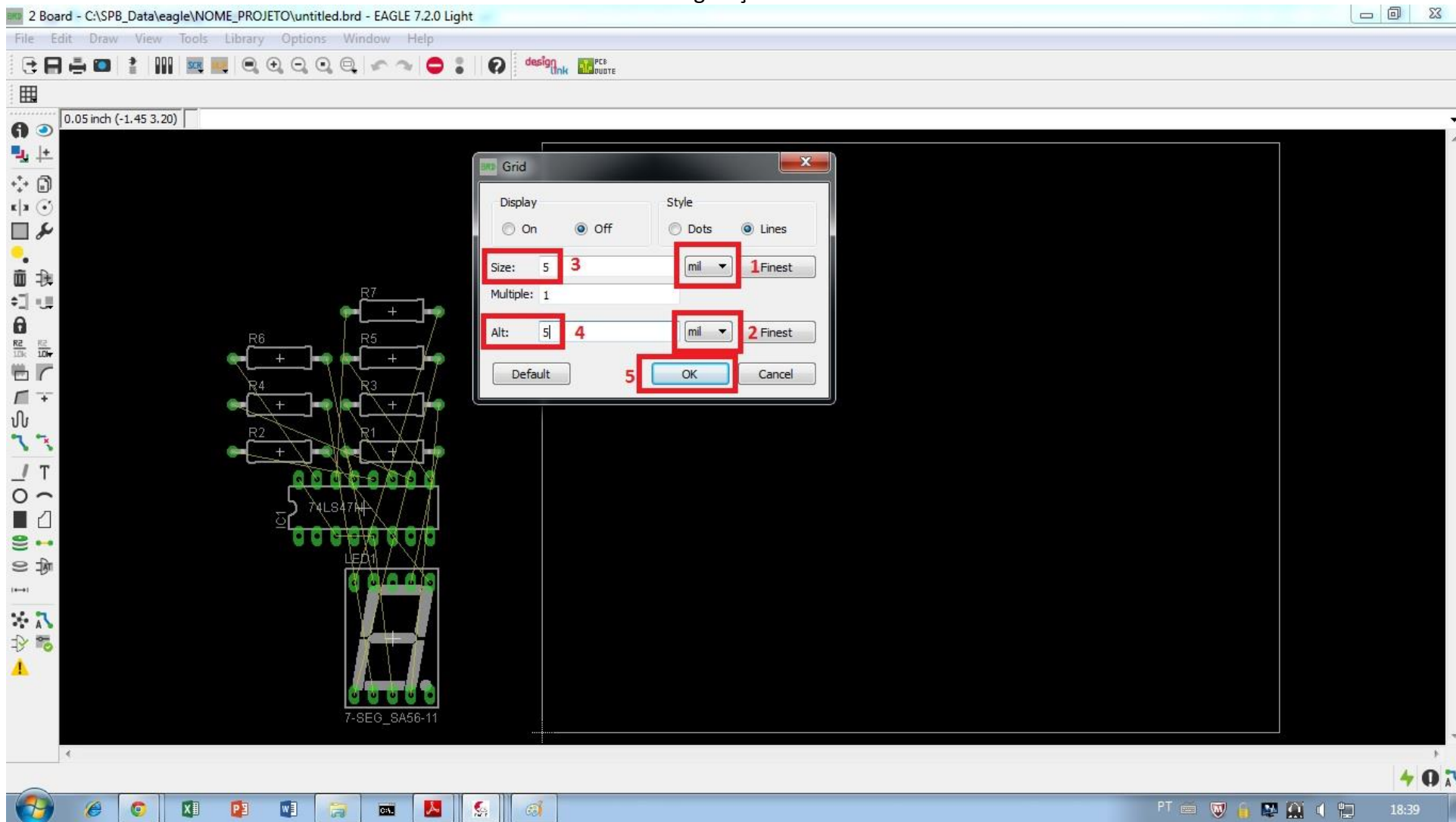


## 17 – Configurações do GRID.

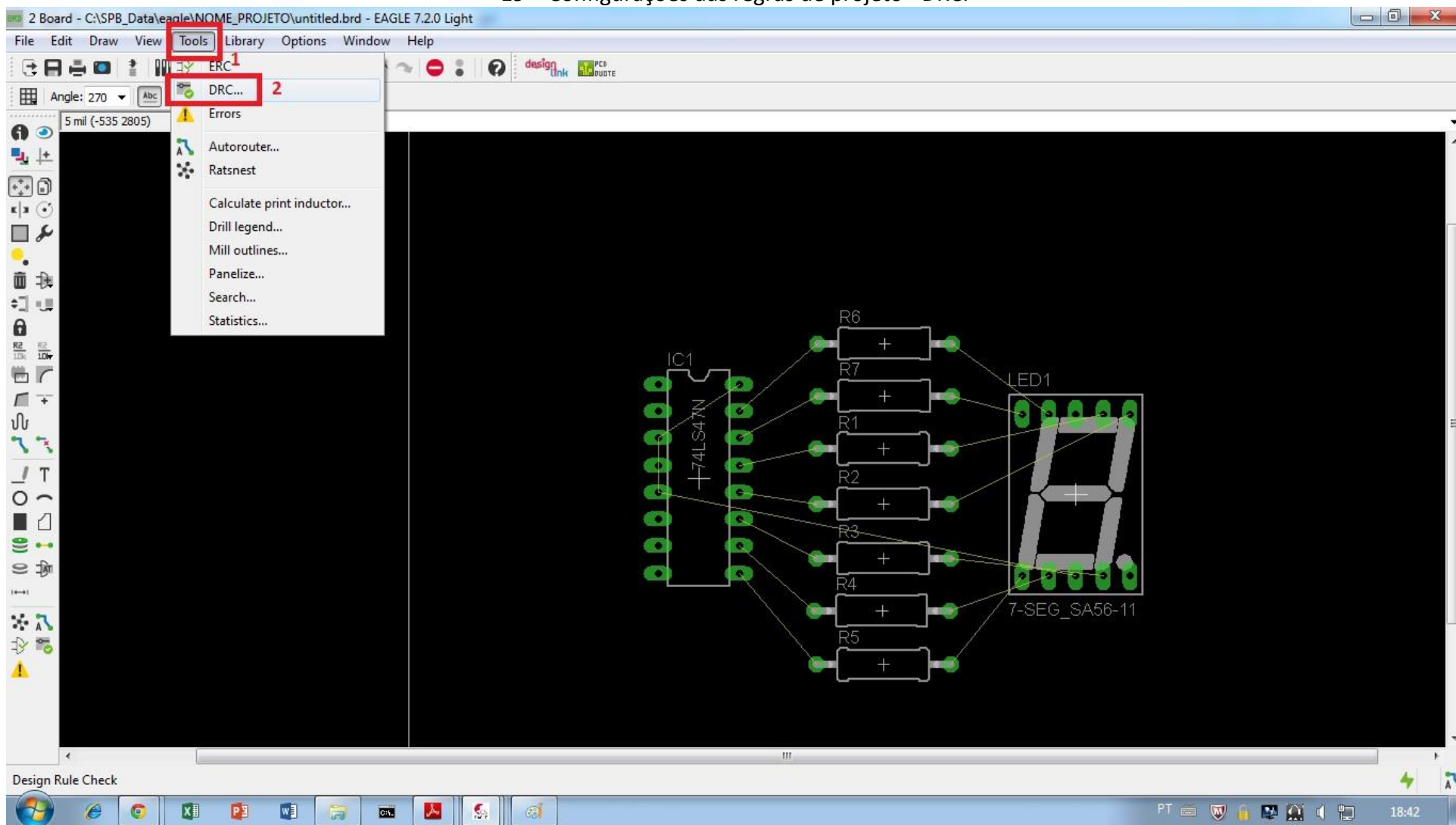




## 18 – Configurações do GRID.



## 19 – Configurações das regras de projeto - DRC.



## 20 – Configurações das regras de projeto – DRC CLEARANCE.

The screenshot shows the Eagle 7.2.0 Light interface with the DRC (default) dialog box open. The dialog box has several tabs: File, Layer, Clearance, Distance, Sizes, Restricting, Shapes, Supply, Masks, and Misc. The Clearance tab is selected and highlighted with a red box labeled '1'. Inside the Clearance tab, there are two sections: 'Different Signals' and 'Same Signals'. The 'Different Signals' section contains a table of settings for Wire, Pad, and Via, with values set to 20mil. The 'Same Signals' section contains settings for Smd, Pad, and Via, also set to 20mil. A red box labeled '2' highlights the entire settings table. To the left of the table is a diagram showing a red rectangular pad and a green circular via with a central hole, and a dimension line indicating the clearance between them. Below the table, there is explanatory text: 'Minimum Clearance between objects in signal layers.', 'The Same Signals check between Smd and Via does not apply to Micro Vias.', 'The Same Signals check does not apply if an Smd and Smd/Pad are in the same package.', and 'Setting the values for the Same Signals checks to 0 disables the respective check.' A red box labeled '3' highlights the 'Apply' button at the bottom right of the dialog. A large red text overlay in the center of the dialog reads 'TROCAR TODOS OS VALORES PARA 20 MILS'. The background shows a PCB layout with a red pad and a green via.

2 Board - C:\SPB\_Data\ eagle\NOME\_PROJETO\untitled.brd - EAGLE 7.2.0 Light

File Edit Draw View Tools Library Options Window Help

Angle: 270

5 mil (-710 2805)

DRC (default)

File Layer **Clearance** Distance Sizes Restricting Shapes Supply Masks Misc

**1**

**Different Signals** **2**

Wire	20mil	Pad	
Pad	20mil	20mil	Via
Via	20mil	20mil	20mil

**Same Signals**

Smd	20mil	20mil	Via
-----	-------	-------	-----

**3**

Minimum Clearance between objects in signal layers.

The Same Signals check between Smd and Via does not apply to Micro Vias.

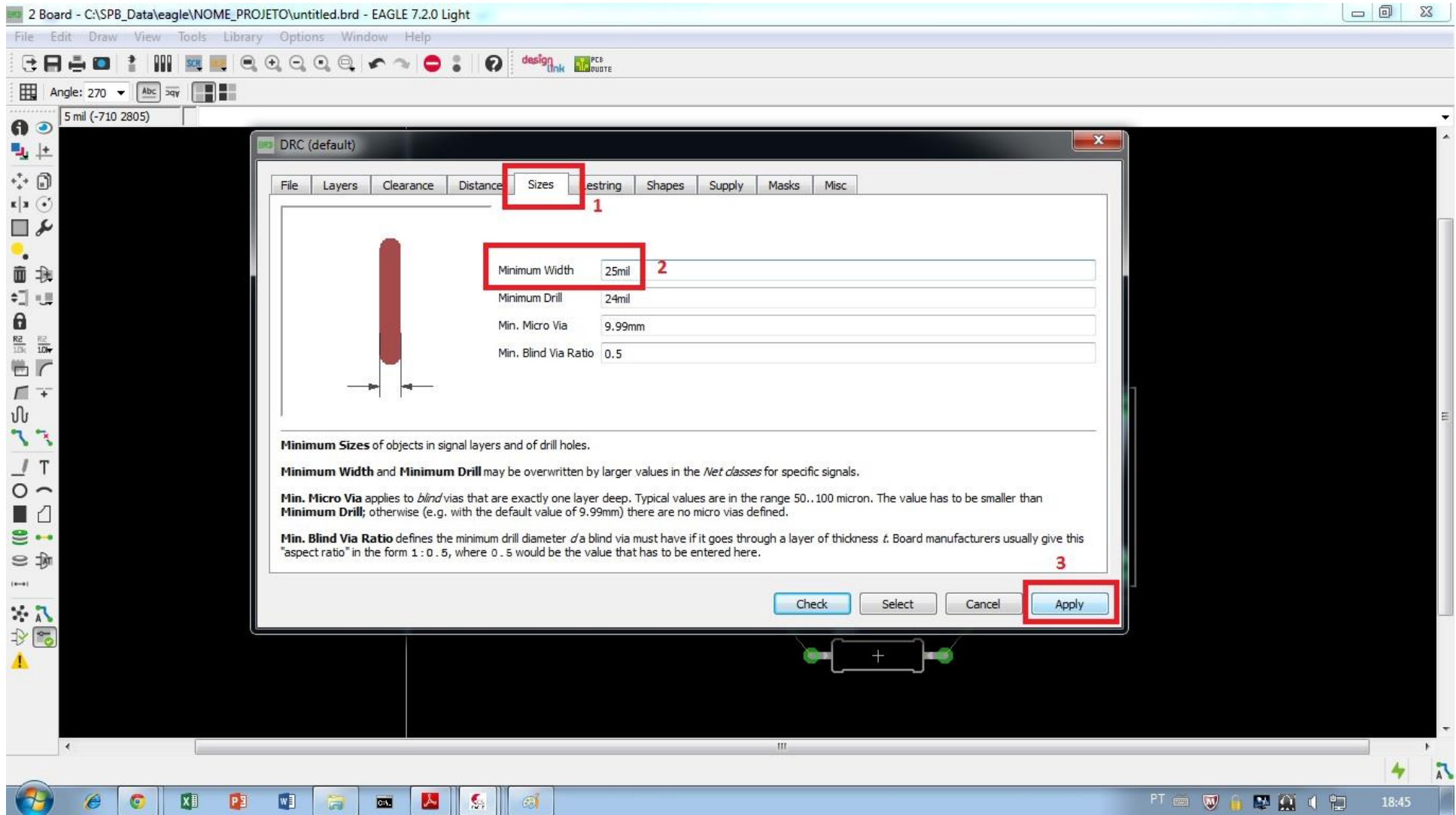
The Same Signals check does not apply if an Smd and Smd/Pad are in the same package.

Setting the values for the Same Signals checks to 0 disables the respective check.

**TROCAR TODOS OS VALORES PARA 20 MILS**

Check Select Cancel **Apply**

## 21 – Configurações das regras de projeto – DRC SIZES.



## 22 – Configurações das regras de projeto – DRC PADS.

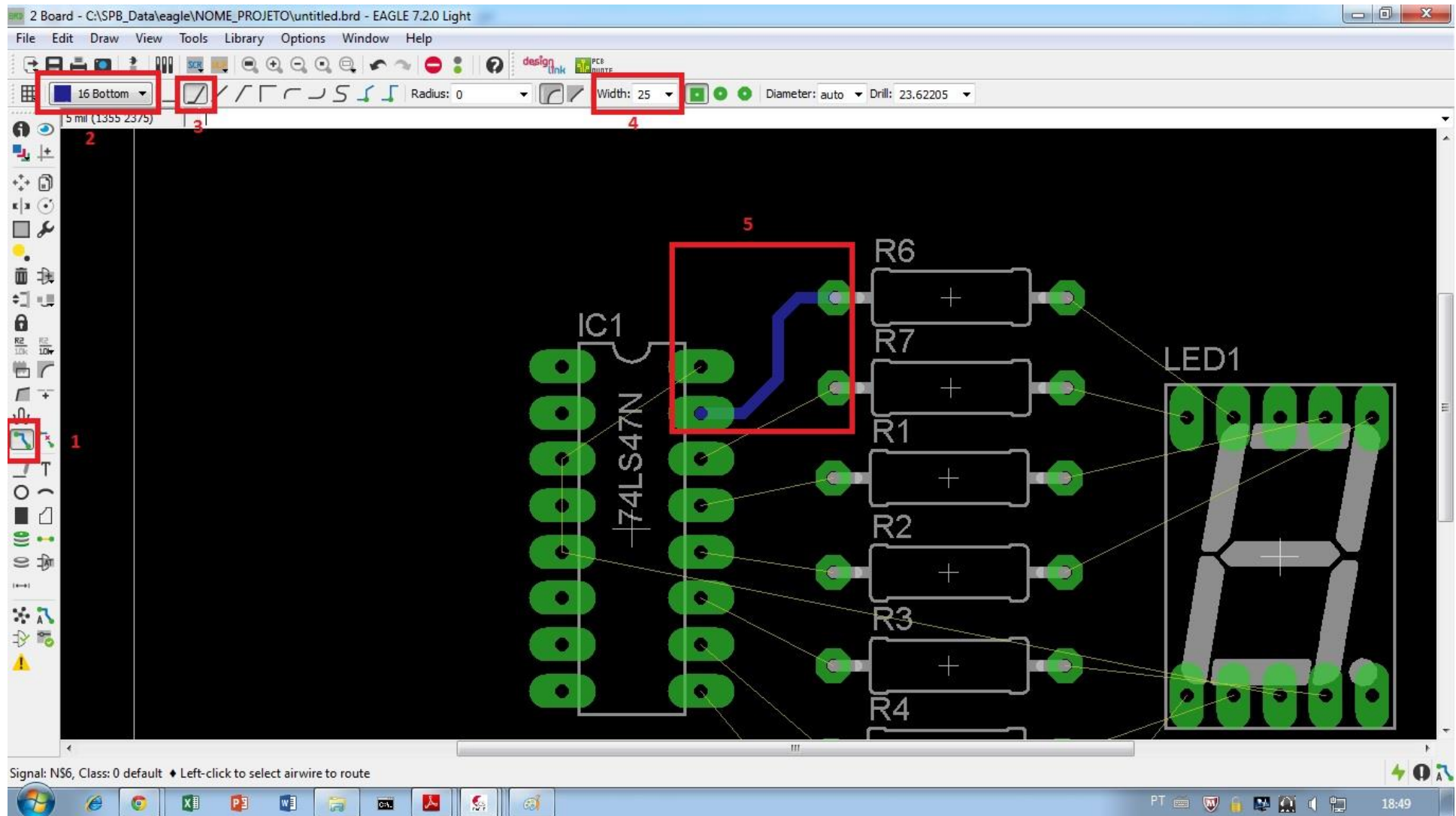
The screenshot shows the Eagle 7.2.0 Light DRC dialog box. The 'Restrings' tab is selected and highlighted with a red box labeled '1'. The 'Pads' section contains a table with the following data:

		Min	%	Max	Diameter
Pads	Top	20mil	25	20mil	<input type="checkbox"/>
	Inner	20mil	25	20mil	<input type="checkbox"/>
	Bottom	20mil	25	20mil	<input type="checkbox"/>
Vias	Outer	8mil	25	20mil	<input type="checkbox"/>
	Inner	8mil	25	20mil	<input type="checkbox"/>
Micro Vias	Outer	4mil	25	20mil	<input type="checkbox"/>
	Inner	4mil	25	20mil	<input type="checkbox"/>

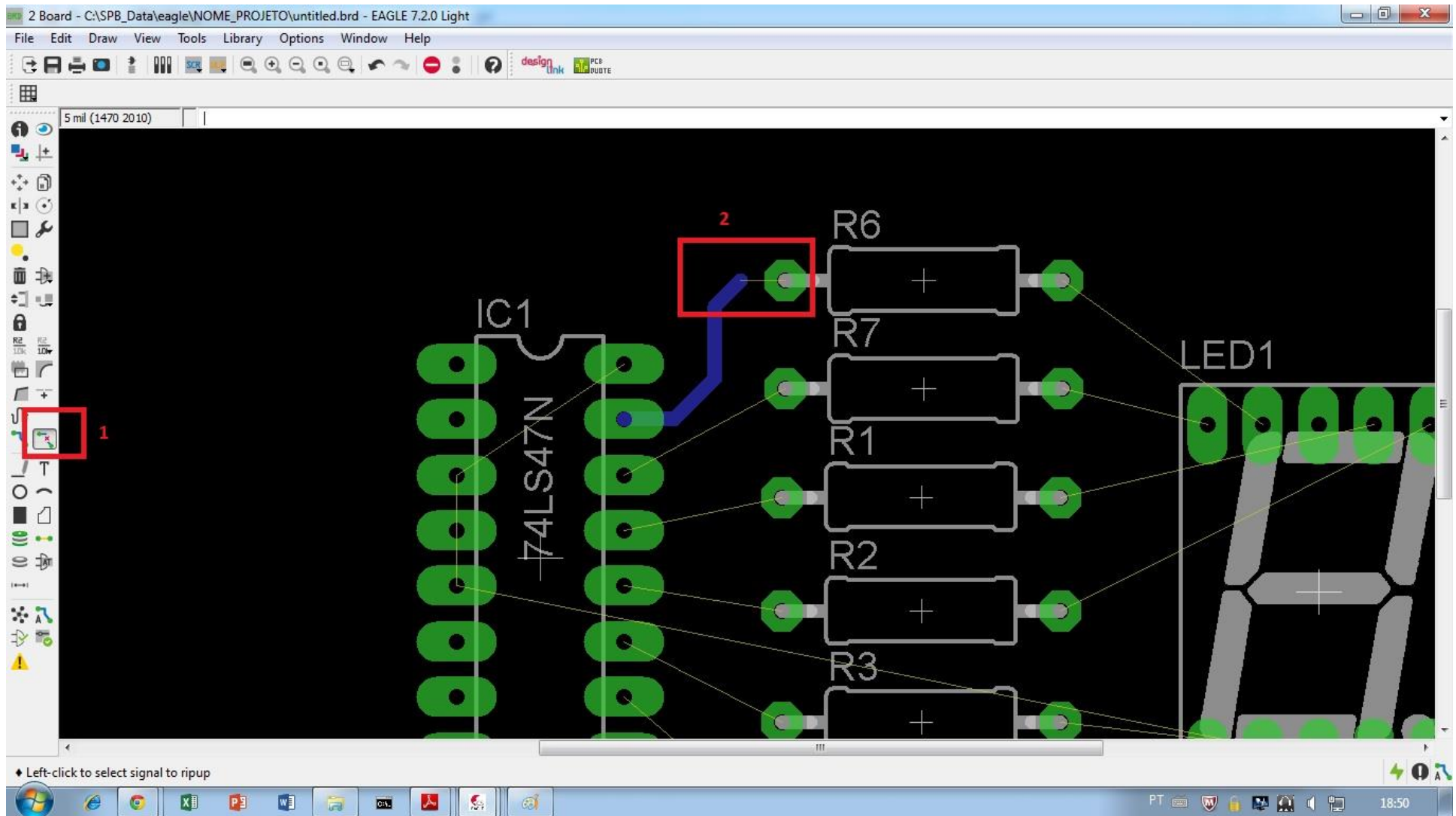
Below the table, there is explanatory text: 'Restrings for pads and vias are defined in percent of the drill diameter (limited by Min and Max). If the diameter of an actual pad or via would result in a larger restring, that value will be used in the outer layers. If the Diameter option is checked the actual pad or via diameter will be taken into account in the inner layers, too. Micro Vias are blind vias that are exactly one layer deep and have a drill diameter that is smaller than the Minimum Drill value defined under Sizes (which may be overwritten by a larger Drill value in the Net classes).'

At the bottom of the dialog, the 'Select' button is highlighted with a red box labeled '4', and the 'Apply' button is highlighted with a red box labeled '3'. The 'Check' button is also visible.

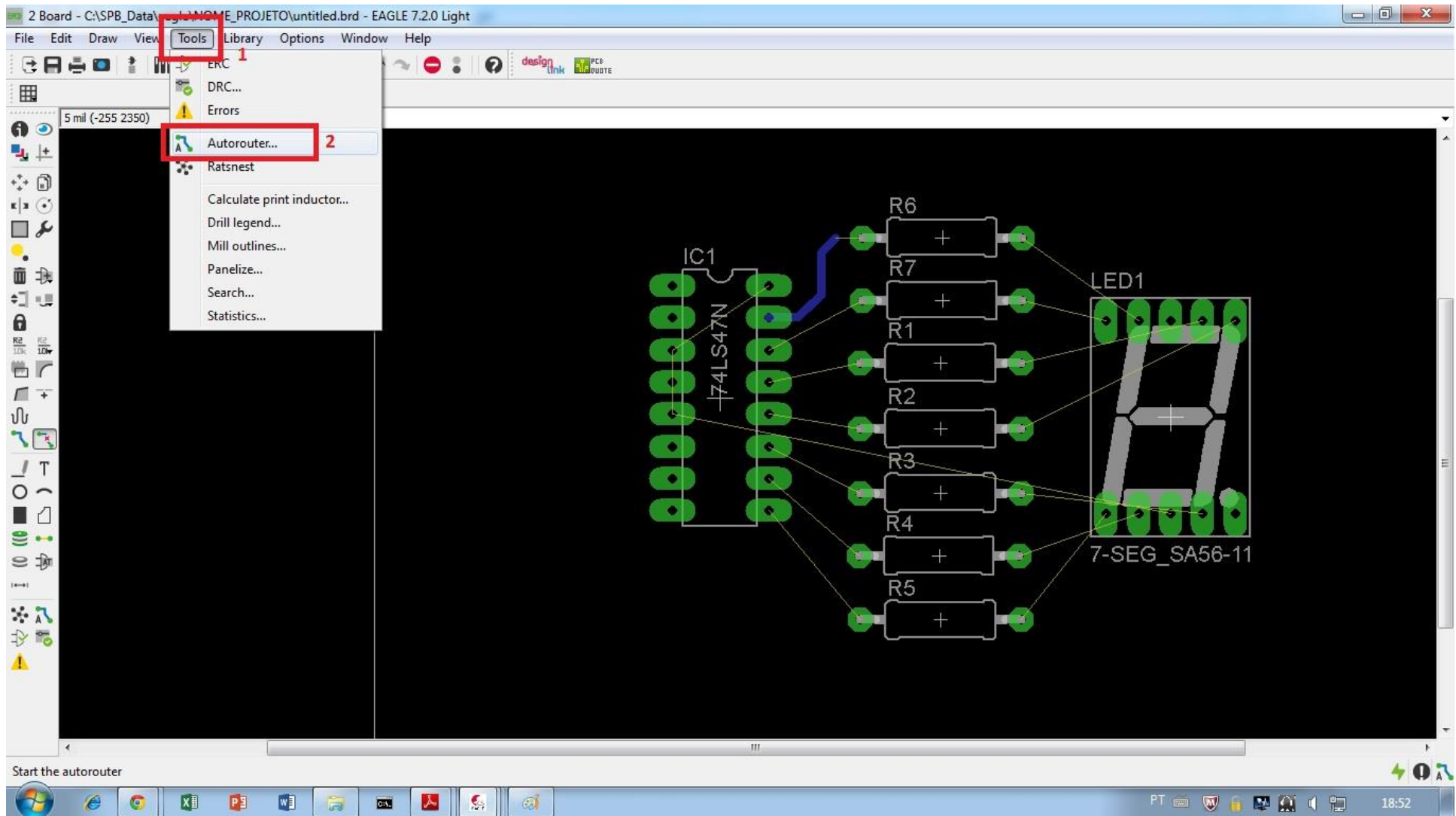
### 23 – Ferramenta de roteamento manual.



24 – Ferramenta para “desrotear” um pedaço de trilha.

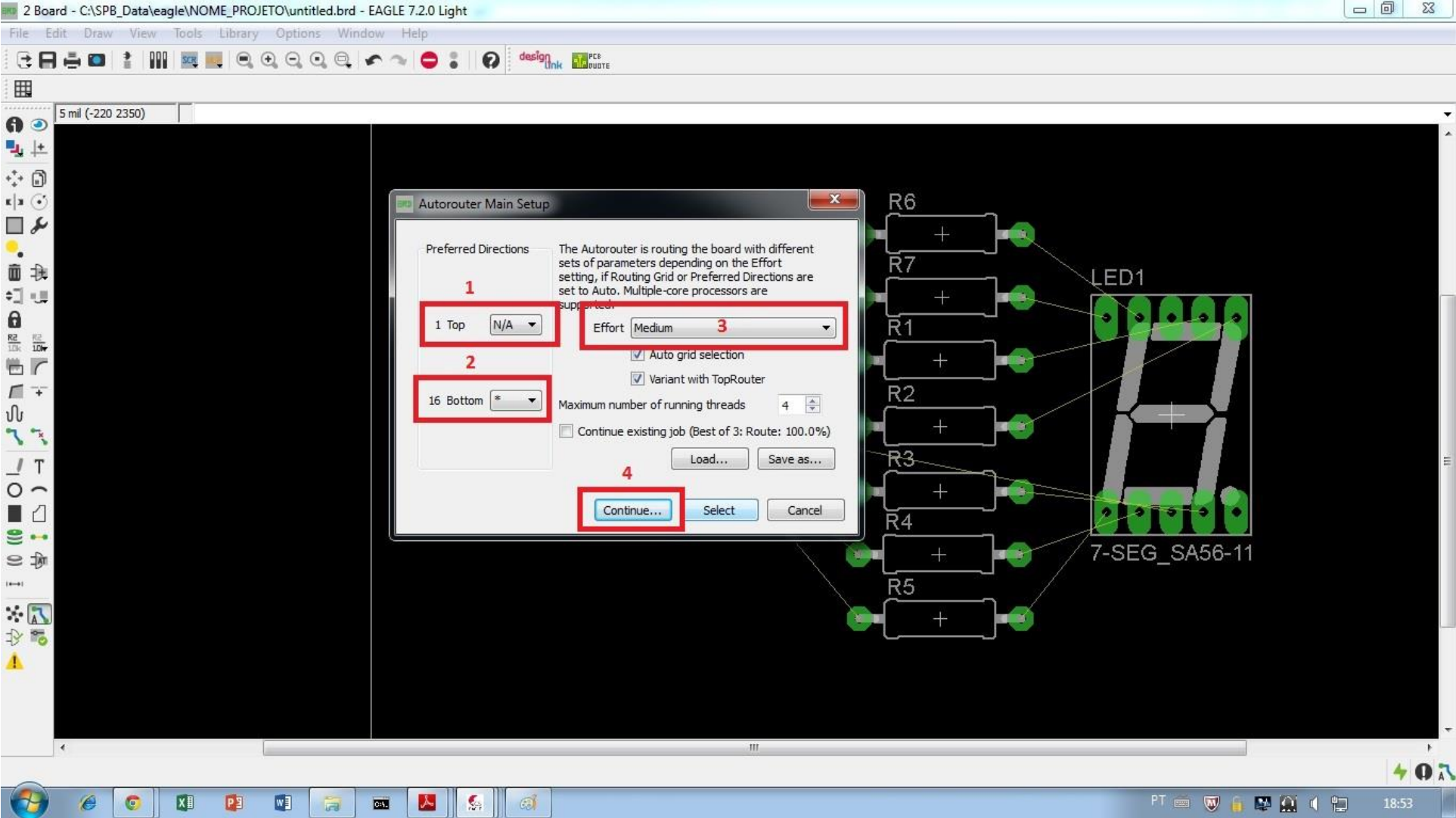


## 25 – Configuração do Autorouter.

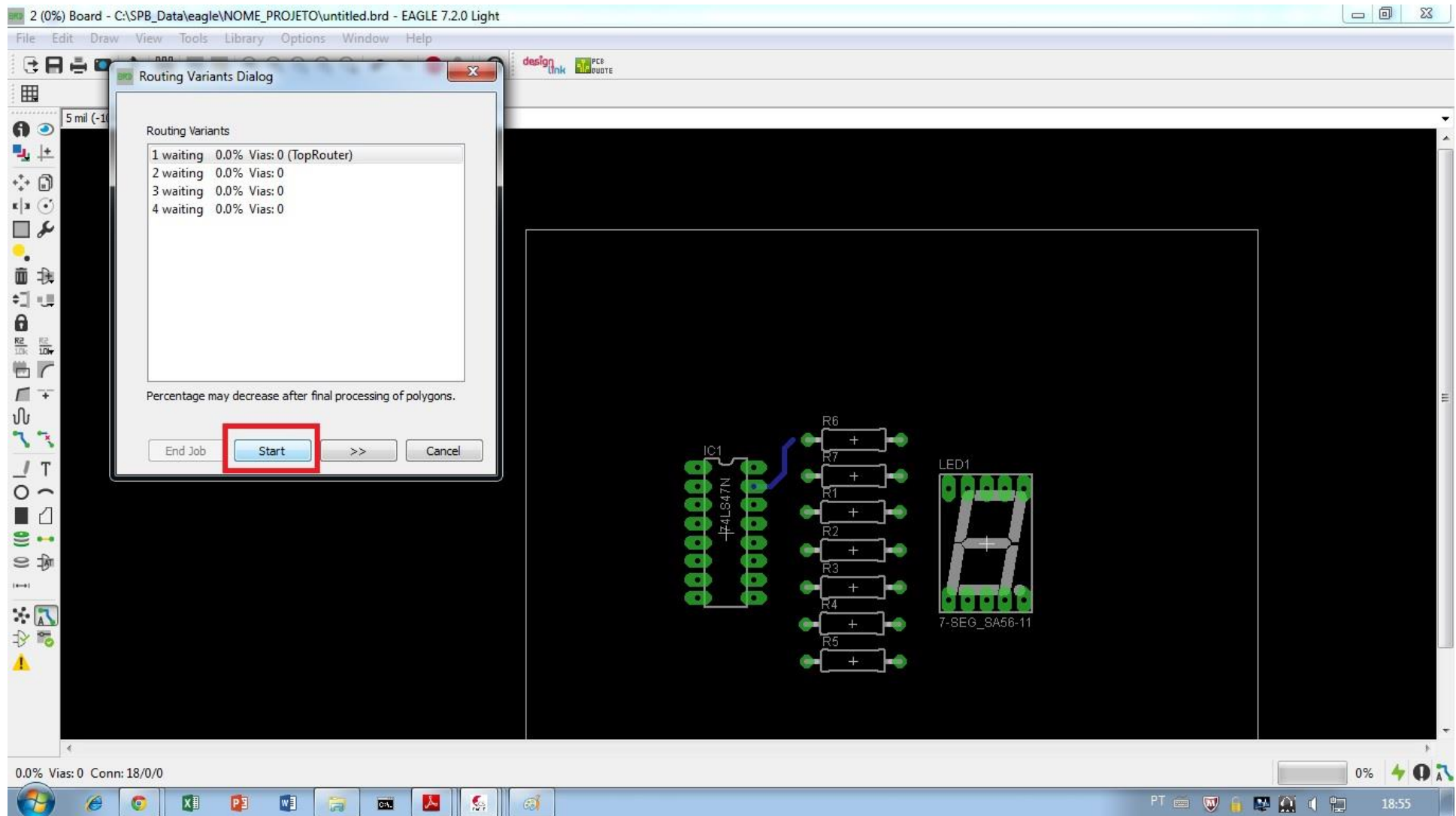




26 – Configuração do Autorouter.



## 27 – Iniciando o auto roteamento.



## 28 – Auto roteamento completo.

The screenshot displays the Eagle PCB software interface. The main window shows a PCB layout with components labeled IC1, R1 through R7, and LED1. The status bar at the bottom indicates "Optimize8: 100.0% Vias: 0 Conn: 18/18/0 Signals: 16/16".

The "Routing Variants Dialog" box is open, showing the following routing variants:

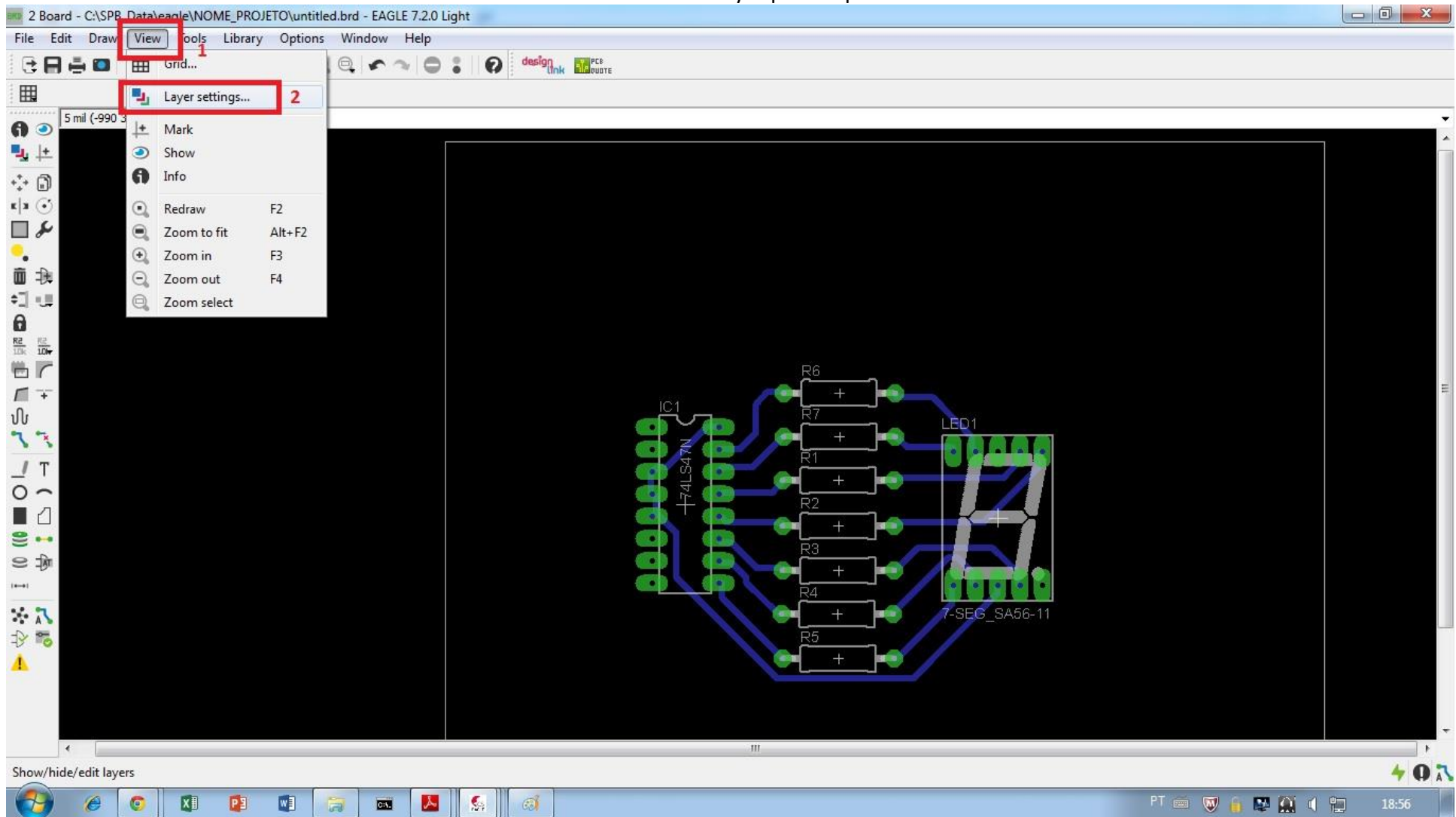
- 1 completed Optimize8: 100.0% Vias: 0 (TopRouter)
- 2 completed Optimize4: 100.0% Vias: 0
- 3 completed Optimize4: 100.0% Vias: 0
- 4 completed Optimize4: 100.0% Vias: 0

The "Evaluate" button is highlighted with a red box. A red box also highlights the first variant. A red number "1" is next to the third variant, and a red number "2" is below the "Evaluate" button.

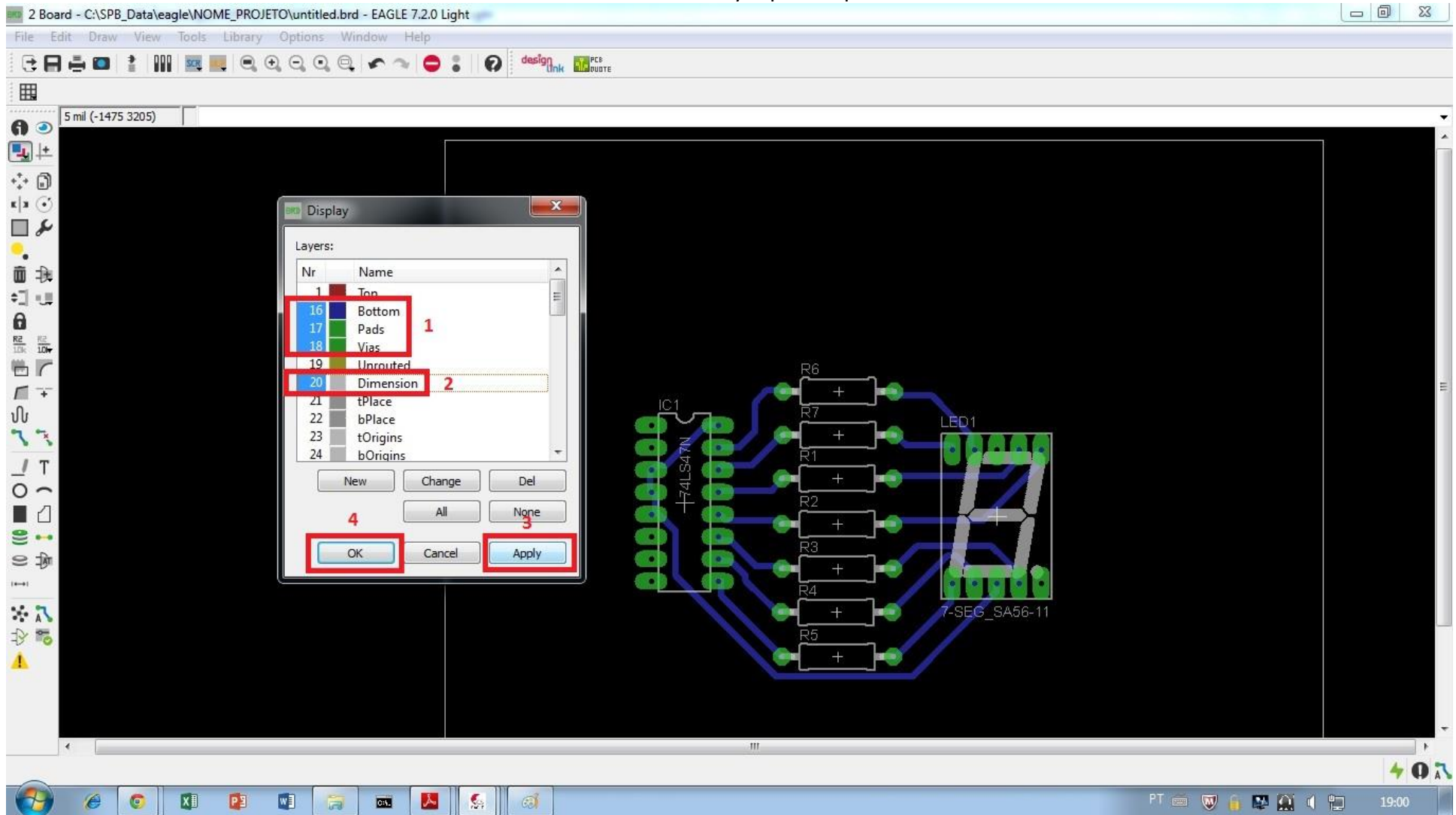
Percentage may decrease after final processing of polygons.

End Job Evaluate >> Cancel

## 29 – Selecionando layer para impressão.



### 30 – Seleccionando layer para impressão.



## ETAPAS PARA CONFECÇÃO DAS PLACAS

1. CONFECÇÃO DO ESQUEMÁTICO.
2. CONFECÇÃO DO LAYOUT.
3. IMPRESSÃO DO LAYOUT EM PAPEL NORMAL.
4. PEDIR NO ALMOXARIFADO PAPEL TRANSFER E RECORTAR EM UM TAMANHO LIGEIRAMENTE MAIOR QUE AS BORDAS DA PLACA E COLAR COM DUREX EM CIMA DA IMPRESSÃO NO PAPEL NORMAL.
5. COLOCAR NA IMPRESSORA NA BANDEJA MANUAL COM O PAPEL TRANSFER VIRADO PARA CIMA E MANDAR IMPRIMIR O MESMO LAYOUT. A IMPRESSÃO DEVE FICAR EM CIMA DO PAPEL TRANSFER.
6. PEDIR NO ALMOXARIFADO UMA PLACA FENOLITE FACE SIMPLES E SERRA. A PLACA DEVERÁ SER CORTADA DO TAMANHO DO LAYOUT.
7. LIXAR COM BOMBRIL O LADO DO COBRE DA PLACA DE FENOLITE PARA RETIRAR A OXIDAÇÃO.
8. COLAR O PAPEL TRANSFER COM A IMPRESSÃO VIRADA PARA O LADO DO COBRE DA PLACA COM DUREX.
9. PEDIR PARA OS LABORATORISTAS LIGAREM A PRENSA E PEDIR PARA ELES PRENSAREM A PLACA.
10. APÓS REALIZADA A PRENSA (DEMORA UNS 10 MINUTOS) RETIRAR COM CUIDADO O PAPEL TRANSFER.
11. FAZER UM FURO EM ALGUM CANTO DA PLACA E PRENDER UM FIO DE MAIS OU MENOS 30cm.
12. LEVAR NO LABORATÓRIO DO SUBSOLO A DIREITA NA PORTA COM GRADE E COLOCAR A PLACA DENTRO DA BACIA COM PERCLORETO DE FERRO PARA CORROER A PLACA. A PLACA DEVE FICAR TOTALMENTE IMERSA COM O FIO PARA FORA PARA SE PODER RETIRAR **NÃO TOCAR NO LÍQUIDO**

**PODE QUEIMAR.** ESTE PROCESSO PODE DEMORAR DE 40 MINUTOS À 1 HORA DEPENDENDO SE O LÍQUIDO ESTÁ NOVO OU NÃO. AGITANDO PELO FIO ACELERA O PROCESSO.

**13.** A PLACA ESTARÁ PRONTA QUANDO NÃO SE ENXERGAR MAIS O COBRE.

**14.** FAZER A FURAÇÃO DA PLACA COM BROCA DE 0,8mm OU DE 1mm NA FURADEIRA DREMEL QUE TEMOS NO ALMOXARIFADO.

**15.** E POR FIM SOLDAR OS COMPONENTES

