

Welcome to OrCAD at Maker Faire! Use this instruction guide to create a schematic in OrCAD Capture. If additional help is needed, view our Capture Tutorial Videos indicated for each section. View the Smart PDF for additional information about the schematic. Throughout the instruction guide, short cut keys are shown after the command in **bold** as well as in the list below.

### Helpful Keyboard Shortcuts

Copy ( <b>CTRL-C</b> )	Place Part- ( <b>P</b> )
End Mode- ( <b>ESC</b> )	Place Power- ( <b>F</b> )
Paste ( <b>CTRL-V</b> )	Place Wire/Bus at Angle ( <b>Hold SHIFT</b> )
Place Bus- ( <b>B</b> )	Rotate- ( <b>R</b> )
Place Ground- ( <b>G</b> )	Select Multiple Items ( <b>Hold CTRL</b> )
Place Net Alias- ( <b>N</b> )	Undo- ( <b>CTRL-Z</b> )
Place No Connect- ( <b>X</b> )	Wire- ( <b>W</b> )

## Section 1: Open and Save the Design

1. Open OrCADforMakerFaire.opj
2. Save your project

File → Save As → Name\_MakerFaire

3. Open your schematic

Double Click on Page 1 in the Project Directory

## Section 2: Place a Title Block (Capture Tutorial 1)

1. Fill in the information for your title block

Options → Design Template → Title Block Tab

Library Name = Path to MAKERFAIRE.OLB

Title Block Name = Titleblock

**\*See Figure 1**

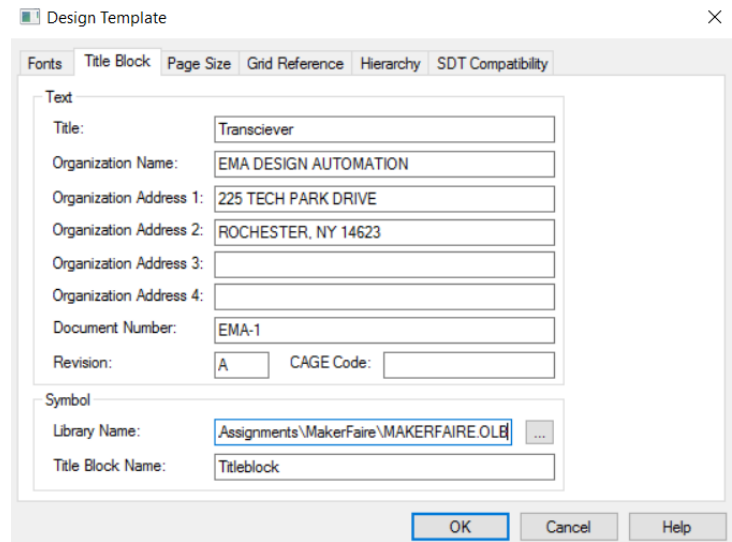
2. Place the Title Block

Place → Title Block → Add Library → Select MakerFaire from Libraries → Place "Titleblock"

Click to place

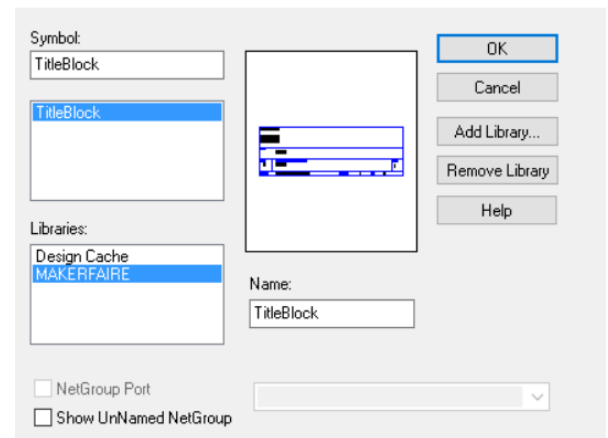
Right Mouse click and select End Mode (ESC)

**\*See Figure 2 and 3**



The screenshot shows the 'Design Template' dialog box with the 'Title Block' tab selected. The 'Text' section contains the following fields: Title (Transceiver), Organization Name (EMA DESIGN AUTOMATION), Organization Address 1 (225 TECH PARK DRIVE), Organization Address 2 (ROCHESTER, NY 14623), Organization Address 3 (empty), Organization Address 4 (empty), Document Number (EMA-1), Revision (A), and CAGE Code (empty). The 'Symbol' section contains: Library Name (Assignments\MakerFaire\MAKERFAIRE.OLB) and Title Block Name (Titleblock). Buttons for OK, Cancel, and Help are at the bottom right.

Figure 1: Title Block Information



The screenshot shows the 'Place Title Block' dialog box. The 'Symbol' field contains 'TitleBlock'. The 'Libraries' list shows 'Design Cache' and 'MAKERFAIRE'. The 'Name' field contains 'TitleBlock'. There are checkboxes for 'NetGroup Port' and 'Show UnNamed NetGroup'. Buttons for OK, Cancel, Add Library..., Remove Library, and Help are on the right.

Figure 2: Place Title Block



Figure 3: Title Block Placement

### Section 3: Placing Parts (Capture Tutorial 3)

Place Parts- View part location and associated schematic symbol in Smart PDF

#### 1. Place → Parts (P)

Select the MAKERFAIRE library and place parts according to the schematic

Select the place part icon or double click on the part to place

Click the schematic to Place

Rotate parts as needed (R)

Right click to End Mode (ESC)

*\*See Figure 4*

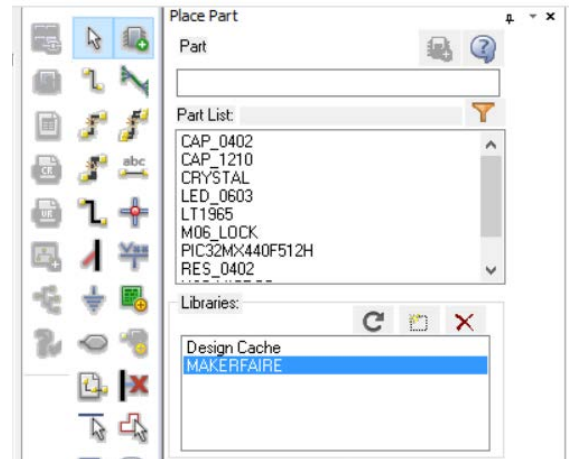


Figure 4: Place Parts

#### 2. Place Ground (G)

Place → Ground → select MakerFaire library → Place GND symbols

Right click to End mode (ESC)

*\*See Figure 5*

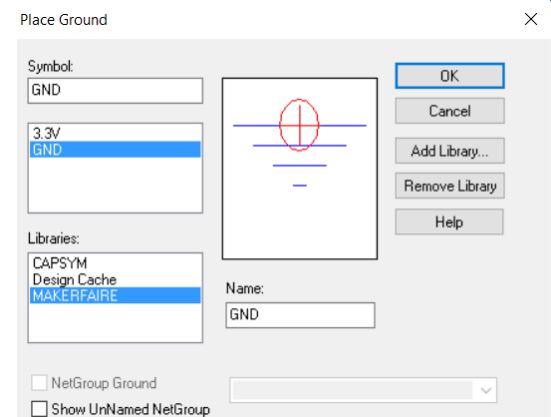


Figure 5: Place Ground

#### 3. Place Power (F)

Place → Power → select MakerFaire library → Place 3.3V symbols

Right Click to End Mode (ESC)

*\*See Figure 6*

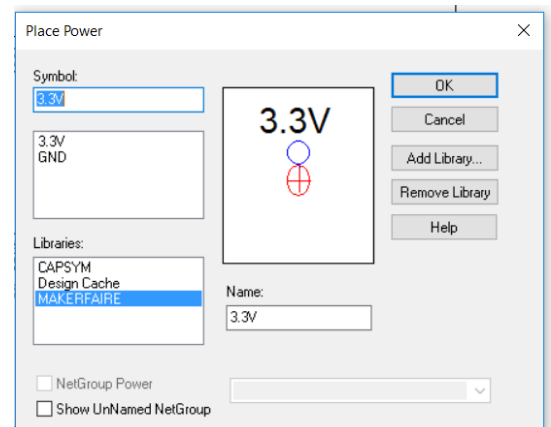


Figure 6: Place Power

*\*Notes: To move parts, select the part(s) and drag.*

*Parts can be rotated after placement by right clicking and selecting rotate.*

*You can copy and paste parts on the schematic (CTRL-C, CTRL-V)*

## Section 4: Annotating the Design (Capture Tutorial 8)

1. Save the design
2. Go back to the project directory and select the design file *\*See Figure 7*

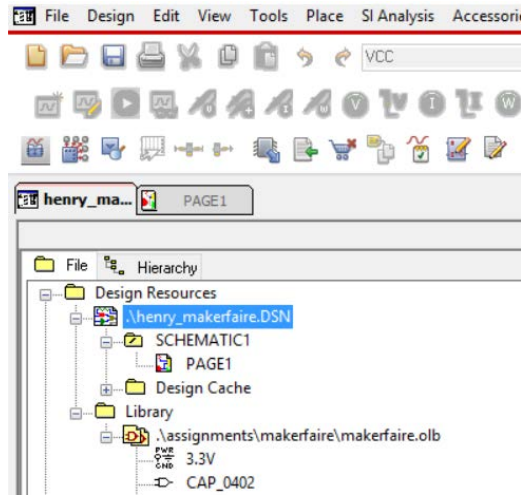


Figure 7: Design Selection

3. Tools → Annotate → Use the settings below in Figure 8

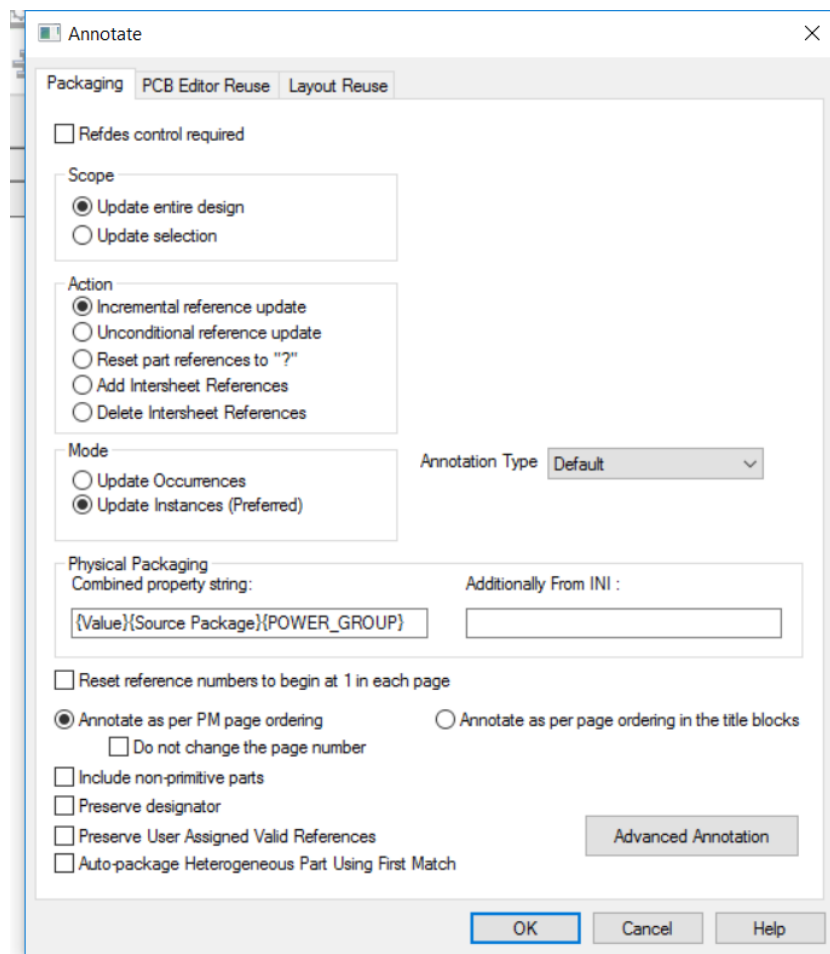


Figure 8: Annotation Settings

## Section 5: Wiring (Capture Tutorial 4)

View Smart PDF for Pin Connections

### 1. Wire Buses: Place → Bus (B)

Click to place the bus

To place bus at an angle, hold down the **SHIFT** key

Right Click to End Mode (**ESC**)

### 2. Connect to Bus: Place → Auto Wire → Connect to Bus

Click the net(s) and then the bus to connect

Add Net names

Right Click to End Mode (**ESC**)

**\*See Figure 9**

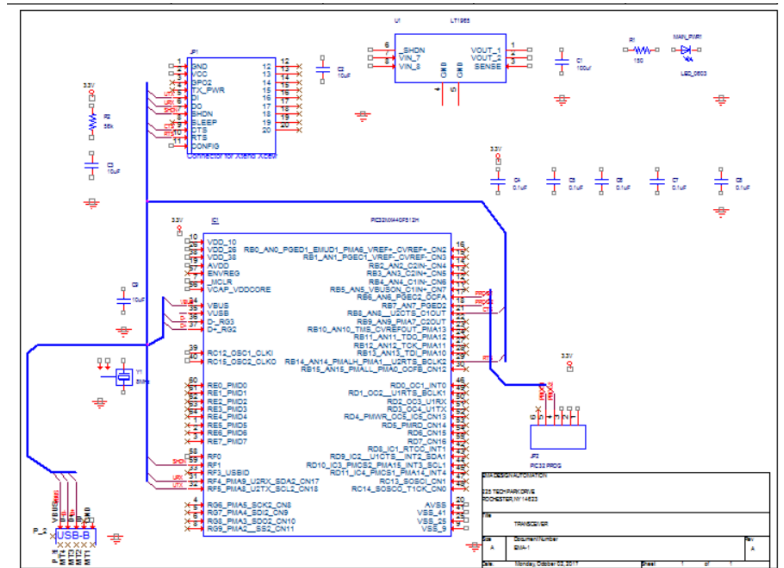


Figure 9: Completed Buses

*\*Notes: You can also connect to bus by adding a wire to the desired pin and placing bus entry (E) to connect. If needed rotate the bus entry (R)*

*If some nets are not named, this can be done when assigning Net Aliases.*

*Auto Wire Syntax for single net selection: VBUS*

*Auto Wire Syntax for multiple net selection: PROG[0-1]*

*Some components may need to be moved out of the way to make bus connections.*

### 3. Wire schematic: Place → Wire (W)

Click to connect

Right click to End Mode (**ESC**)

Add wires to: Pins 1 and 3 of JP2

Pin 48 of IC1

Pin 1 of JP1.

Net Aliases will be used to finish these connections.

**\*See Figure 10**

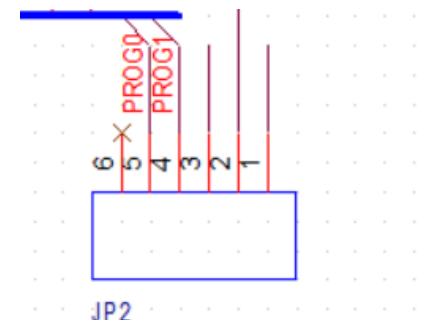


Figure 10: JP2 Connections

### 4. Place No connects: Place → No Connect (X)

Click on pins that do not have connections

Right click to End Mode (**ESC**)

*\*Note: Don't forget to connect Pin 2 of JP1 to the USB bus (Place → Auto wire → Connect to Bus- name VBUS)*

## Section 6: Assigning Net Aliases (Capture Tutorial 5)

1. Add Net Aliases to Wires: Place → Net Alias (N)

Use Net Alias name RESET for: Pin 1 on JP2

Pin 48 on IC1.

Use (R) to rotate, click to place.

Right click and Edit Properties.

Change the Net Alias to GND

Assign GND to: Pin 3 of JP2

Pin 1 of JP1

GND net on the schematic (Example: GND pin of USB-B)

2. Add net aliases to the buses

PROG[0-1]

XCVR[0-4]

USB[0-2]

Right click to End Mode (ESC)

**\*See Figure 11**

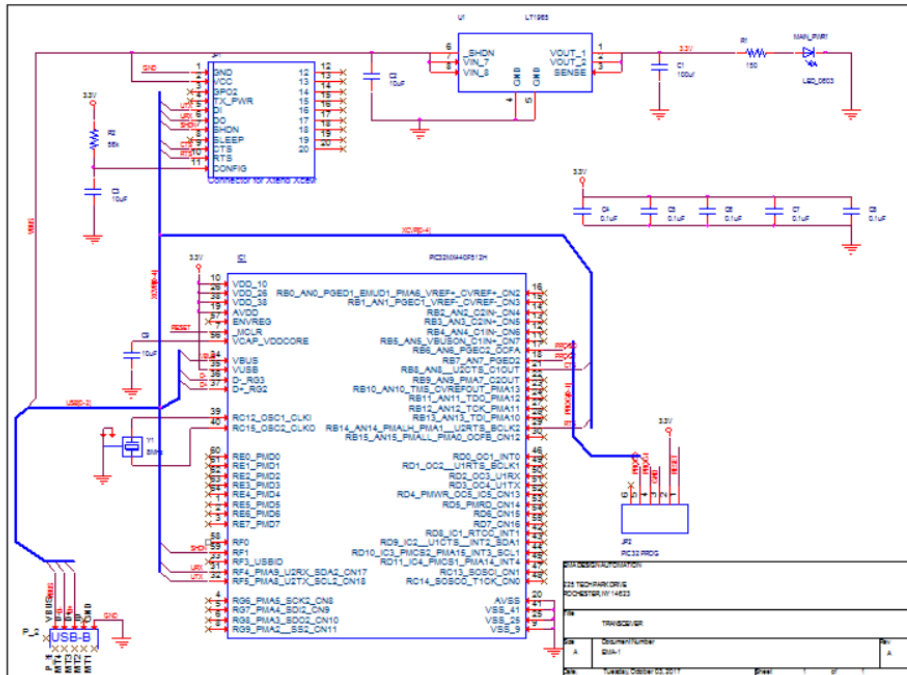


Figure 11: Completed Net Alias Assignment

**\*Note:** By assigning the same Net alias to multiple pins, the physical wire connection does not need to be made.

## Section 7: Component Values (Capture Tutorial 7)

View Smart PDF for Component Values

1. Add Component values for capacitors, resistors, LED and crystal:

Either double click on the value or right click the component → Edit properties

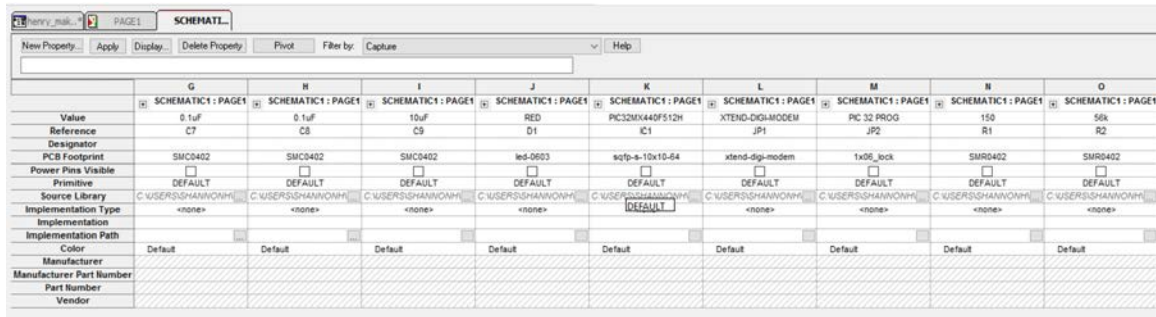
2. To view all of the component properties:

Right click → Selection Filter → Clear All → Check Parts → Ok

Highlight schematic (only parts are selected)

Right click and Edit Properties

**\*See Figure 12**



	G	H	I	J	K	L	M	N	O
Value	0.1uF	0.1uF	10uF	RED	PC32MX440F512H	XTEND-DIGI-MODEM	PC-32 PROG	150	56k
Reference	C7	C8	C9	D1	IC1	JP1	JP2	R1	R2
Designator									
PCB Footprint	SMC0402	SMC0402	SMC0402	led-0603	sqfp-s-10x10-64	xtend-digi-modem	tx06_lock	SMR0402	SMR0402
Power Pins Visible	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Primitive	DEFAULT	DEFAULT	DEFAULT	DEFAULT	DEFAULT	DEFAULT	DEFAULT	DEFAULT	DEFAULT
Source Library	C:\USERS\SHANNON\H\...	C:\USERS\SHANNON\H\...	C:\USERS\SHANNON\H\...	C:\USERS\SHANNON\H\...	C:\USERS\SHANNON\H\...	C:\USERS\SHANNON\H\...	C:\USERS\SHANNON\H\...	C:\USERS\SHANNON\H\...	C:\USERS\SHANNON\H\...
Implementation Type	<none>	<none>	<none>	<none>	DEFAULT	<none>	<none>	<none>	<none>
Implementation Path									
Color	Default	Default	Default	Default	Default	Default	Default	Default	Default
Manufacturer									
Manufacturer Part Number									
Part Number									
Vendor									

Figure 12: All Component Properties

**\*Notes:** Add values to multiple parts by highlighting a selection, right click and edit properties. Right click on Value and choose edit. Add the value and select OK.

Use **CTRL** to select multiple parts.

## Optional Exercises

### Section 8: Assigning Differential Pairs (Capture Tutorial 6)

1. In project directory select the design file  
Tools→Create Differential Pair → Auto Setup

**\*See Figure 13**

2. Fill in the information below:

Prefix: DP

+ Filter: +

- Filter: -

**\*See Figure 14**

3. Click in the Diff Pair window to Auto Generate  
Click Create and Close

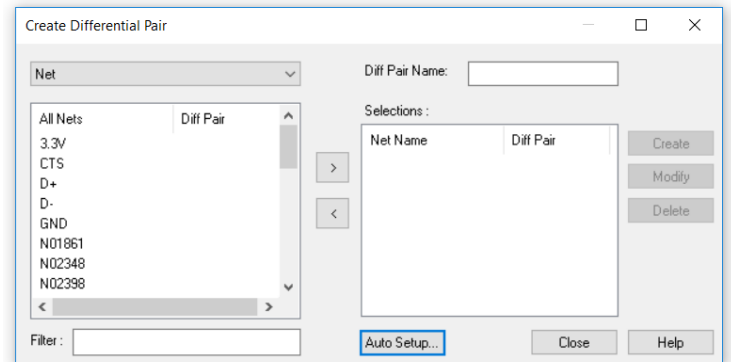


Figure 13: Differential Pair Creation

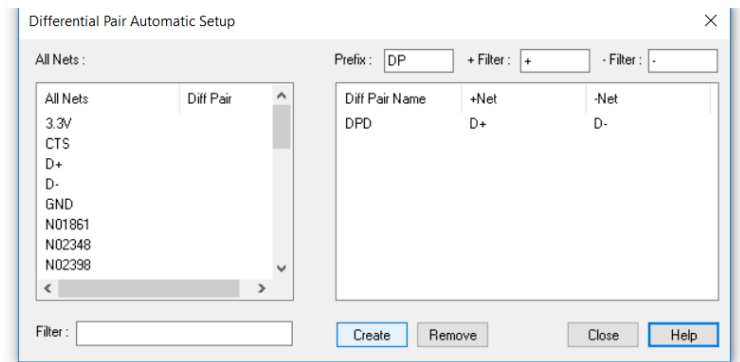


Figure 14: Differential Pair Auto Set up

### Section 9: Smart PDF Generation (Capture Tutorial 11)

1. Select design file from the project directory
2. File→ Export→ PDF
3. Set converter path to the path of the gswin64c.exe file and use the default settings shown in Figure 15
4. Click ok to generate smart PDF

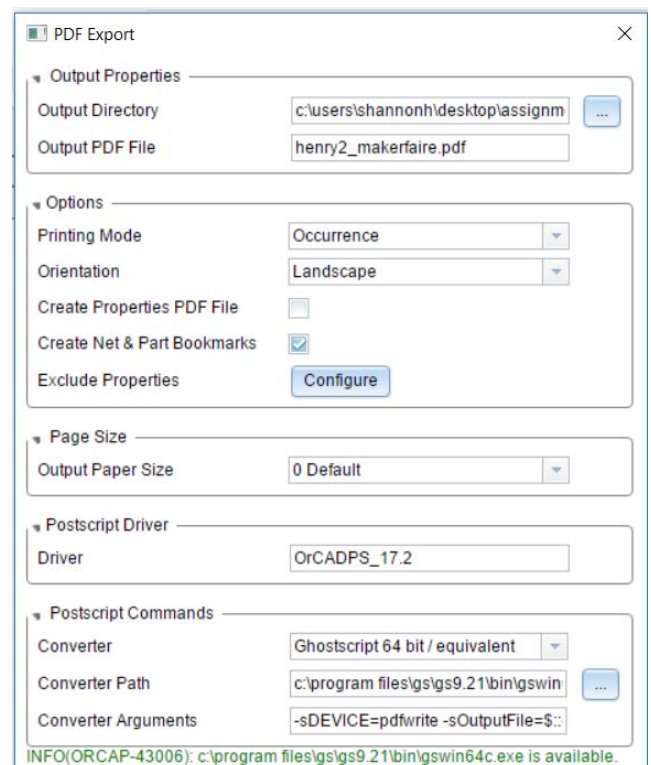


Figure 15: Smart PDF Settings