

processor

review 1 : ok
review 2 :

processor.sch

paula

review 1 : ok
review 2 :

paula.sch

serial

review 1 : ok
review 2 :

serial.sch

power

review 1 : ok
review 2 :

power.sch

chipram

review 1 : ok
review 2 :

chipram.sch

zorro

review 1 : ok
review 2 :

zorro.sch

pc slots

review 1 : ok
review 2 :

pc slots.sch

expansion logic

review 1 : ok
review 2 :

expansion logic.sch

agnus

review 1 : ok
review 2 :

agnus.sch

parallel

review 1 : ok
review 2 :

parallel.sch

denise

review 1 : ok
review 2 :

denise.sch

coprocessor slot

review 1 : ok
review 2 :

coprocessor slot.sch

Notes: XX4256-15 pins 18 and 19 are swapped in the original scanned schematics
pin 18 of U501 is DRD2, and 19 is DRD3.

Rev 6.3 fixes :

- Kickstart 2.03 and Denise 8373

Rev 6.4 fixes :

- Agnus 8375
- Instead of R101, add a capacitor (100nF) from right side of R101 to ground

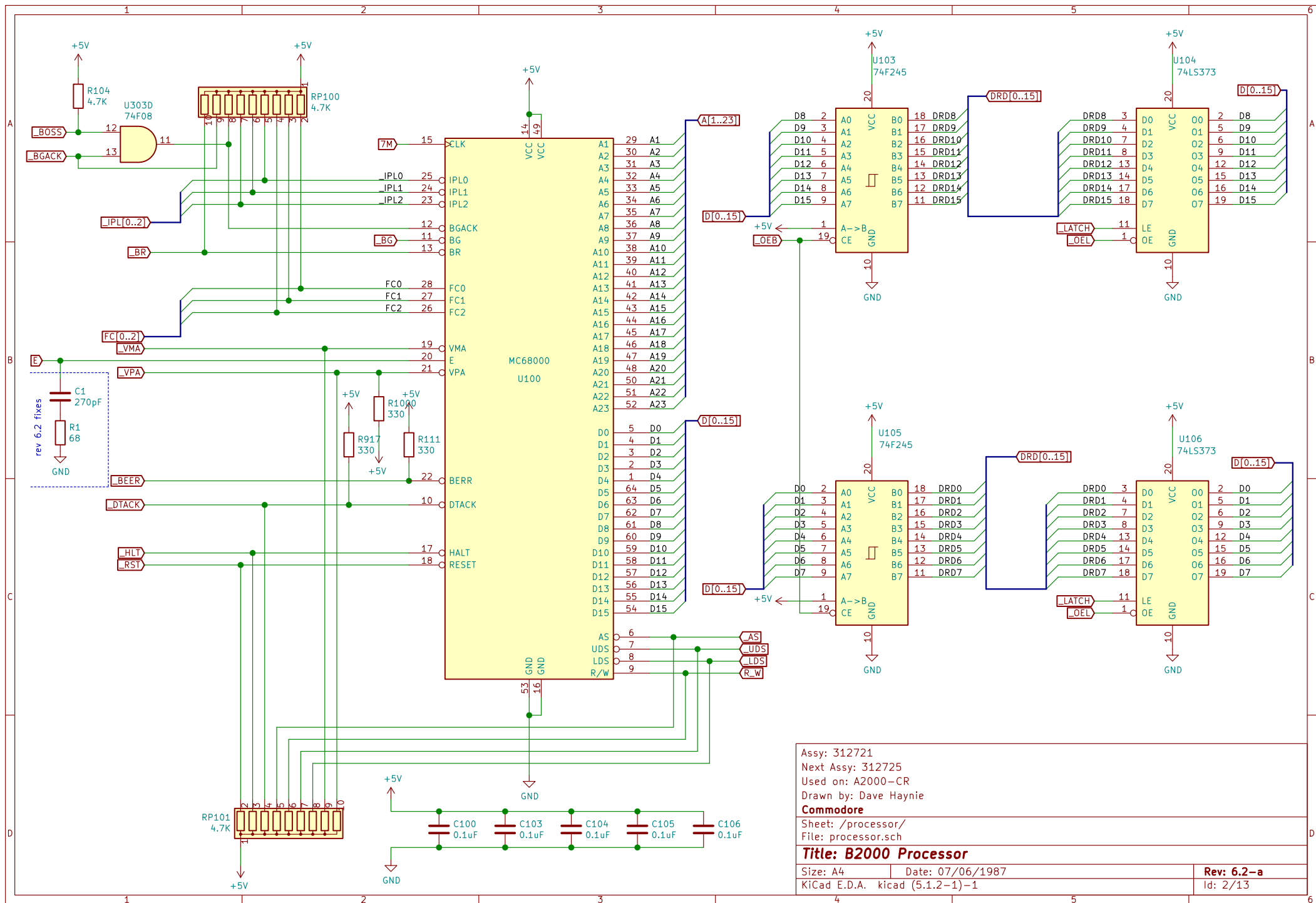
Used on: A2000-CR
Drawn by: Dave Haynie
Commodore

Sheet: /
File: amiga2000.sch

Title:

Size: A4 Date: 07/06/1987
KiCad E.D.A. kicad (5.1.2-1)-1

Rev: 6.2-a
Id: 1/13



Assy: 312721
 Next Assy: 312725
 Used on: A2000-CR
 Drawn by: Dave Haynie

Commodore

Sheet: /processor/
 File: processor.sch

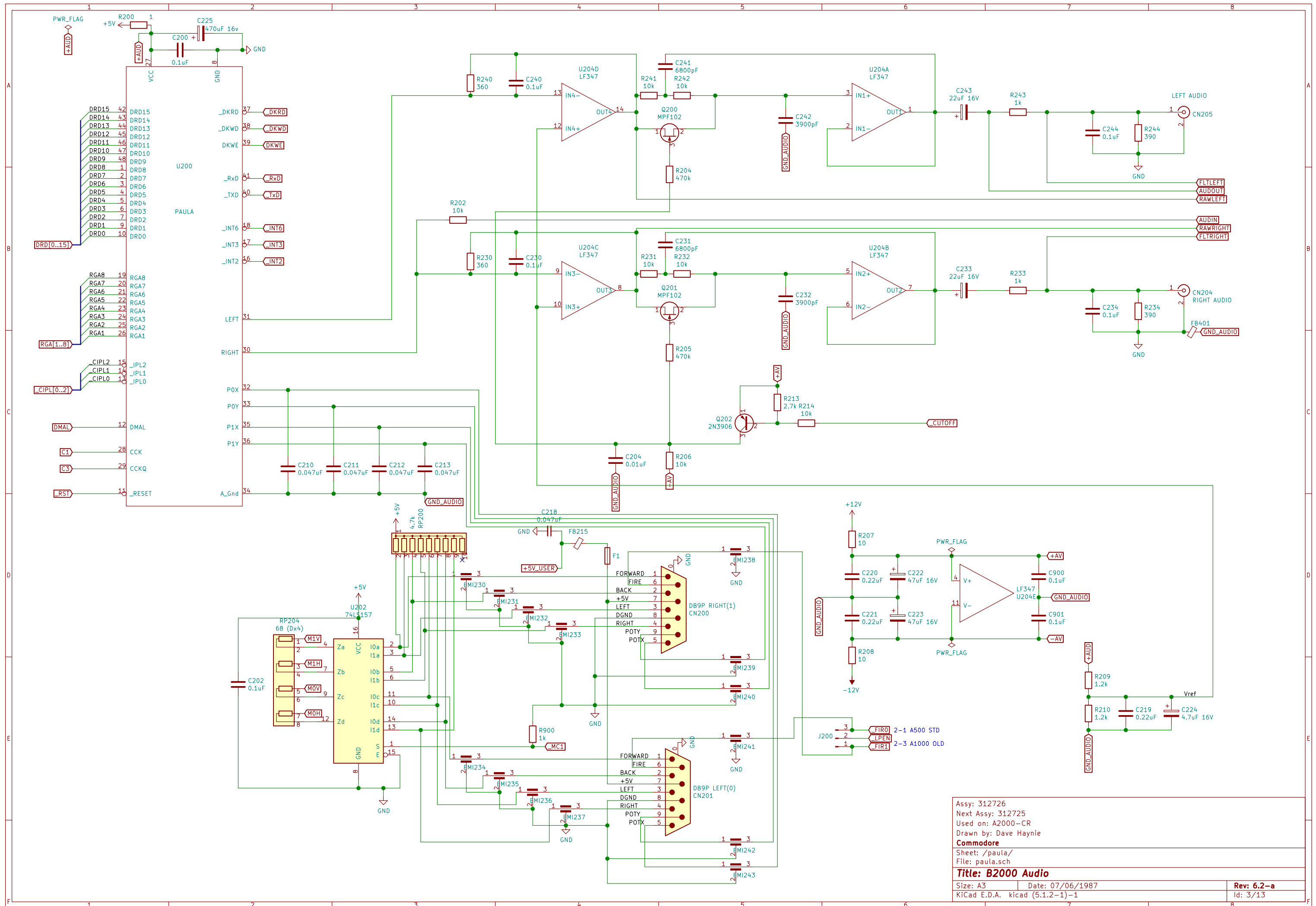
Title: B2000 Processor

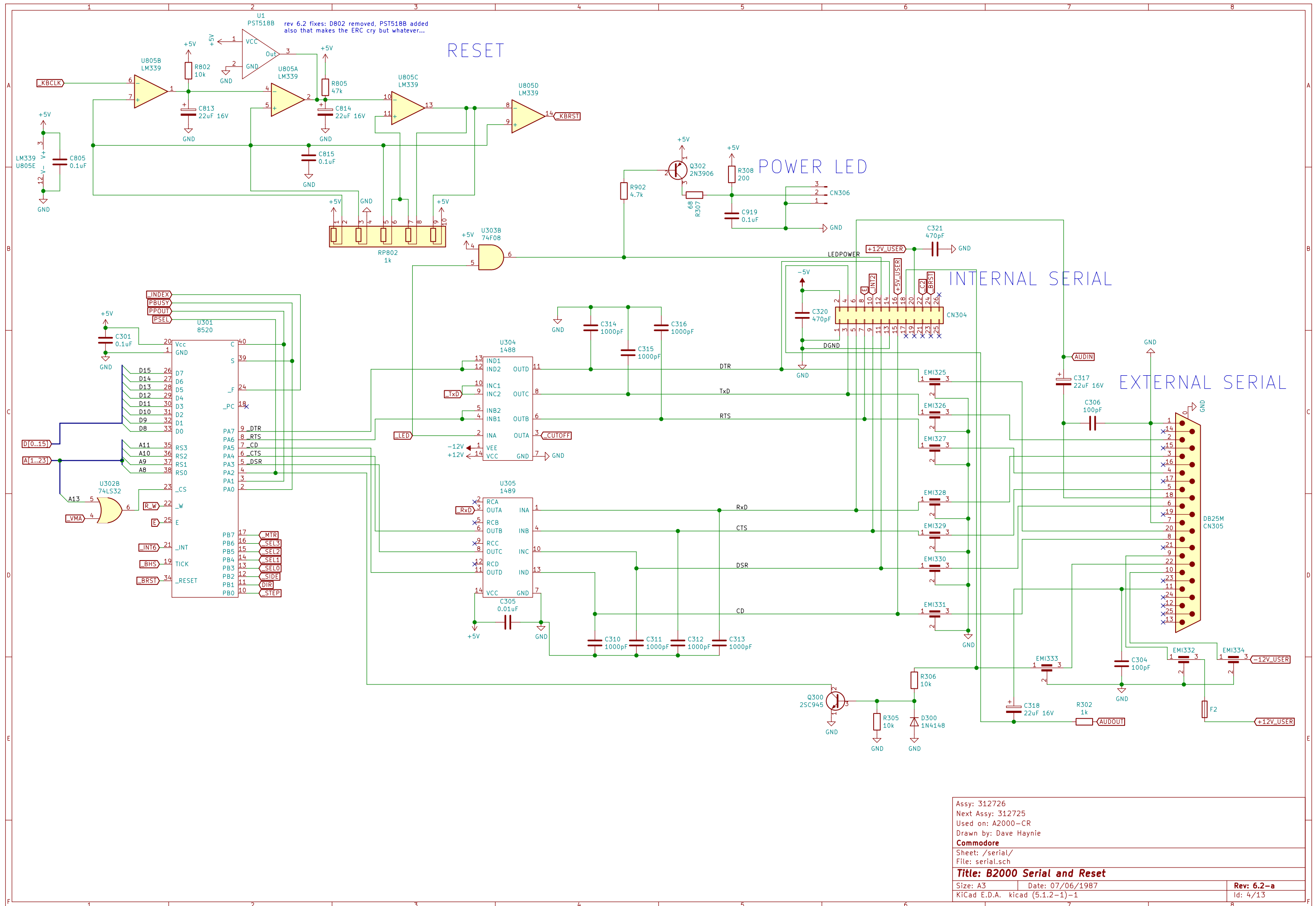
Size: A4 Date: 07/06/1987

KiCad E.D.A. kicad (5.1.2-1)-1

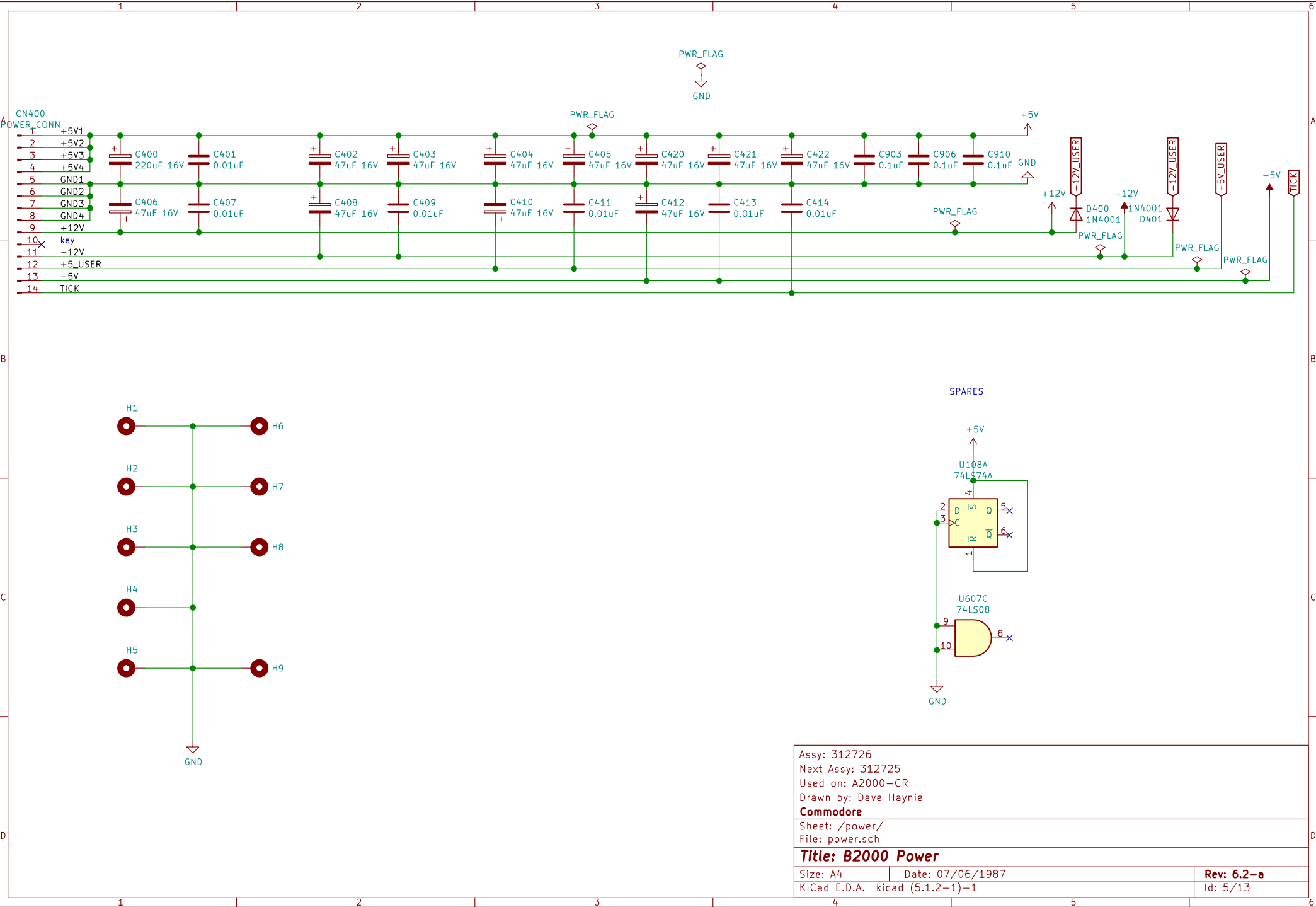
Rev: 6.2-a

Id: 2/13

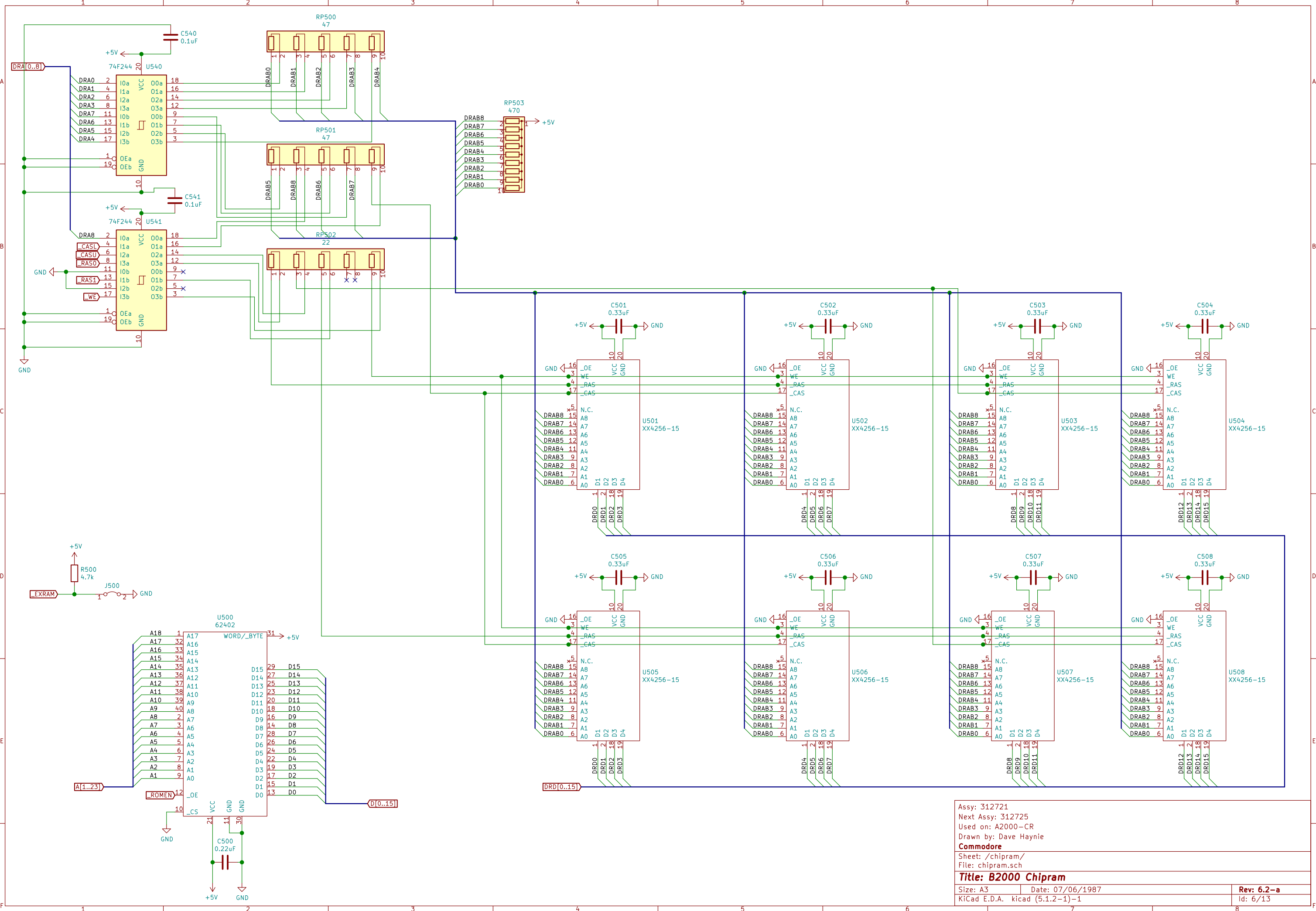




Assy: 312726		
Next Assy: 312725		
Used on: A2000-CR		
Drawn by: Dave Haynie		
Commodore		
Sheet: /serial/		
File: serial.sch		
Title: B2000 Serial and Reset		
Size: A3	Date: 07/06/1987	Rev: 6.2-a
KiCad E.D.A. kicad (5.1.2-1)-1		Id: 4/13



Assy: 312726		
Next Assy: 312725		
Used on: A2000-CR		
Drawn by: Dave Haynie		
Commodore		
Sheet: /power/		
File: power.sch		
Title: B2000 Power		
Size: A4	Date: 07/06/1987	Rev: 6.2-a
KiCad E.D.A. kicad (5.1.2-1)-1		Id: 5/13



Assy: 312721
Next Assy: 312725
Used on: A2000-CR
Drawn by: Dave Haynie

Commodore

Sheet: /chipram/
File: chipram.sch

Title: B2000 Chipram

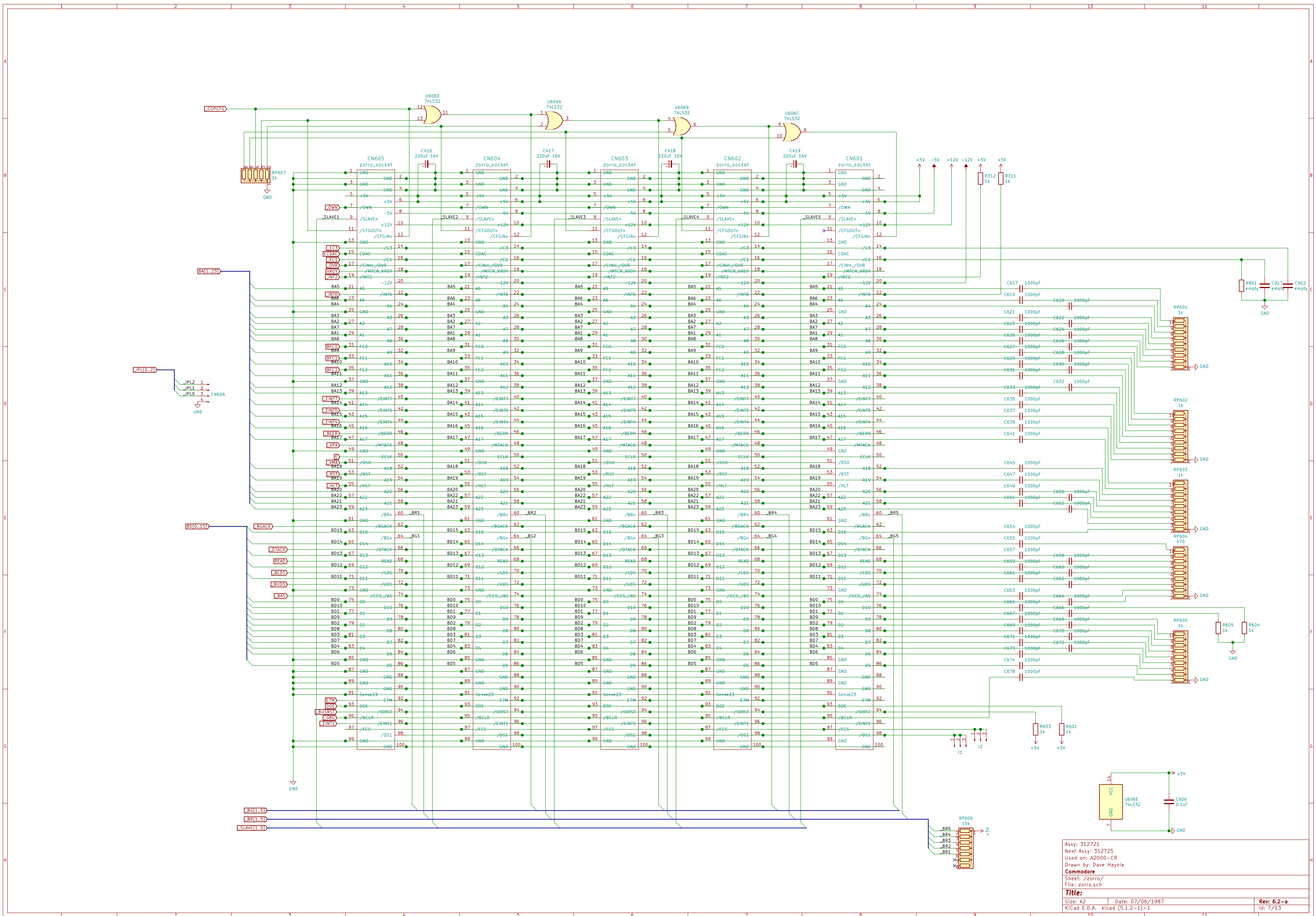
Size: A3

Date: 07/06/1987

Rev: 6.2-a

KiCad E.D.A. kicad (5.1.2-1)-1

Id: 6/13



+12V -12V -5V +5V

C703
0.22uF

C704
0.22uF

C702
0.22uF

C705
0.22uF

C701
22uF 16V

C700
0.22uF

CN700	
B1	2
A1	3
B2	4
A2	5
B3	6
A3	7
B4	8
A4	9
B5	10
A5	11
B6	12
A6	13
B7	14
A7	15
B8	16
A8	17
B9	18
A9	19
B10	20
A10	21
B11	22
A11	23
B12	24
A12	25
B13	26
A13	27
B14	28
A14	29
B15	30
A15	31
B16	32
A16	33
B17	34
A17	35
B18	36
A18	37
B19	38
A19	39
B20	40
A20	41
B21	42
A21	43
B22	44
A22	45
B23	46
A23	47
B24	48
A24	49
B25	50
A25	51
B26	52
A26	53
B27	54
A27	55
B28	56
A28	57
B29	58
A29	59
B30	60
A30	61
B31	62
A31	63

CN701	
B1	2
A1	3
B2	4
A2	5
B3	6
A3	7
B4	8
A4	9
B5	10
A5	11
B6	12
A6	13
B7	14
A7	15
B8	16
A8	17
B9	18
A9	19
B10	20
A10	21
B11	22
A11	23
B12	24
A12	25
B13	26
A13	27
B14	28
A14	29
B15	30
A15	31
B16	32
A16	33
B17	34
A17	35
B18	36
A18	37
B19	38
A19	39
B20	40
A20	41
B21	42
A21	43
B22	44
A22	45
B23	46
A23	47
B24	48
A24	49
B25	50
A25	51
B26	52
A26	53
B27	54
A27	55
B28	56
A28	57
B29	58
A29	59
B30	60
A30	61
B31	62
A31	63

CN702	
B1	2
A1	3
B2	4
A2	5
B3	6
A3	7
B4	8
A4	9
B5	10
A5	11
B6	12
A6	13
B7	14
A7	15
B8	16
A8	17
B9	18
A9	19
B10	20
A10	21
B11	22
A11	23
B12	24
A12	25
B13	26
A13	27
B14	28
A14	29
B15	30
A15	31
B16	32
A16	33
B17	34
A17	35
B18	36
A18	37
B19	38
A19	39
B20	40
A20	41
B21	42
A21	43
B22	44
A22	45
B23	46
A23	47
B24	48
A24	49
B25	50
A25	51
B26	52
A26	53
B27	54
A27	55
B28	56
A28	57
B29	58
A29	59
B30	60
A30	61
B31	62
A31	63

CN703	
B1	2
A1	3
B2	4
A2	5
B3	6
A3	7
B4	8
A4	9
B5	10
A5	11
B6	12
A6	13
B7	14
A7	15
B8	16
A8	17
B9	18
A9	19
B10	20
A10	21
B11	22
A11	23
B12	24
A12	25
B13	26
A13	27
B14	28
A14	29
B15	30
A15	31
B16	32
A16	33
B17	34
A17	35
B18	36
A18	37
B19	38
A19	39
B20	40
A20	41
B21	42
A21	43
B22	44
A22	45
B23	46
A23	47
B24	48
A24	49
B25	50
A25	51
B26	52
A26	53
B27	54
A27	55
B28	56
A28	57
B29	58
A29	59
B30	60
A30	61
B31	62
A31	63

CN704	
D1	1
C1	2
D2	3
C2	4
D3	5
C3	6
D4	7
C4	8
D5	9
C5	10
D6	11
C6	12
D7	13
C7	14
D8	15
C8	16
D9	17
C9	18
D10	19
C10	20
D11	21
C11	22
D12	23
C12	24
D13	25
C13	26
D14	27
C14	28
D15	29
C15	30
D16	31
C16	32
D17	33
C17	34
D18	35
C18	36

CN705	
D1	1
C1	2
D2	3
C2	4
D3	5
C3	6
D4	7
C4	8
D5	9
C5	10
D6	11
C6	12
D7	13
C7	14
D8	15
C8	16
D9	17
C9	18
D10	19
C10	20
D11	21
C11	22
D12	23
C12	24
D13	25
C13	26
D14	27
C14	28
D15	29
C15	30
D16	31
C16	32
D17	33
C17	34
D18	35
C18	36

CN706	
D1	1
C1	2
D2	3
C2	4
D3	5
C3	6
D4	7
C4	8
D5	9
C5	10
D6	11
C6	12
D7	13
C7	14
D8	15
C8	16
D9	17
C9	18
D10	19
C10	20
D11	21
C11	22
D12	23
C12	24
D13	25
C13	26
D14	27
C14	28
D15	29
C15	30
D16	31
C16	32
D17	33
C17	34
D18	35
C18	36

CN707	
D1	1
C1	2
D2	3
C2	4
D3	5
C3	6
D4	7
C4	8
D5	9
C5	10
D6	11
C6	12
D7	13
C7	14
D8	15
C8	16
D9	17
C9	18
D10	19
C10	20
D11	21
C11	22
D12	23
C12	24
D13	25
C13	26
D14	27
C14	28
D15	29
C15	30
D16	31
C16	32
D17	33
C17	34
D18	35
C18	36

C711 1000pF
C713 1000pF
C714 1000pF
C716 1000pF
C717 1000pF
C719 1000pF
C720 1000pF
C722 1000pF

C726 1000pF
C728 1000pF
C730 1000pF
C732 1000pF
C734 1000pF
C736 1000pF
C738 1000pF
C740 1000pF
C742 1000pF
C744 1000pF
C746 1000pF
C748 1000pF
C750 1000pF
C752 1000pF
C754 1000pF
C756 1000pF
C758 1000pF
C790 1000pF
C791 1000pF
C793 1000pF

C794 1000pF
C796 1000pF
C760 1000pF
C762 1000pF
C764 1000pF
C766 1000pF
C768 1000pF
C770 1000pF

C776 1000pF
C778 1000pF
C780 1000pF
C782 1000pF
C784 1000pF
C786 1000pF
C787 1000pF
C789 1000pF

RP700
1k

RP701
1k

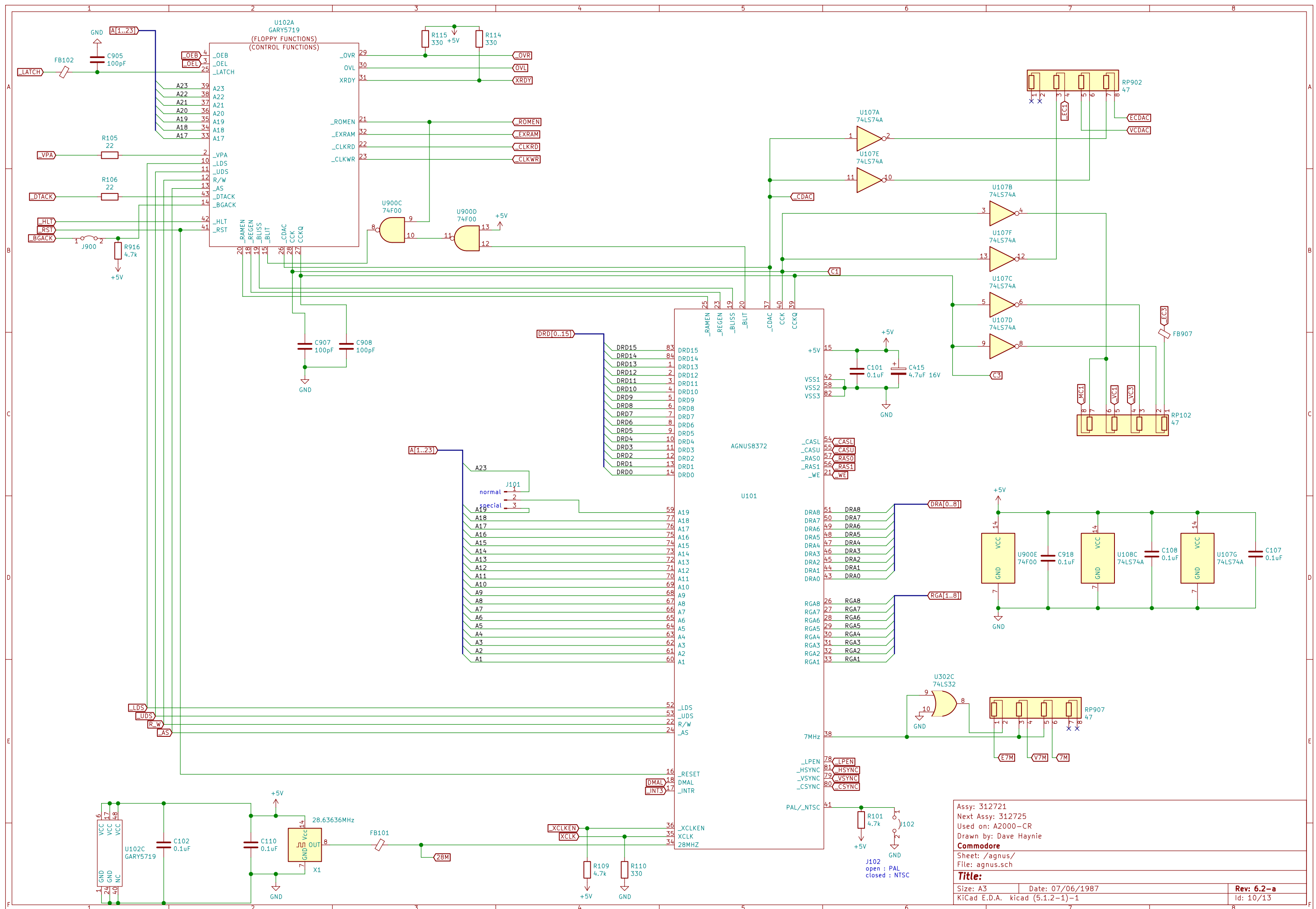
RP702
1k

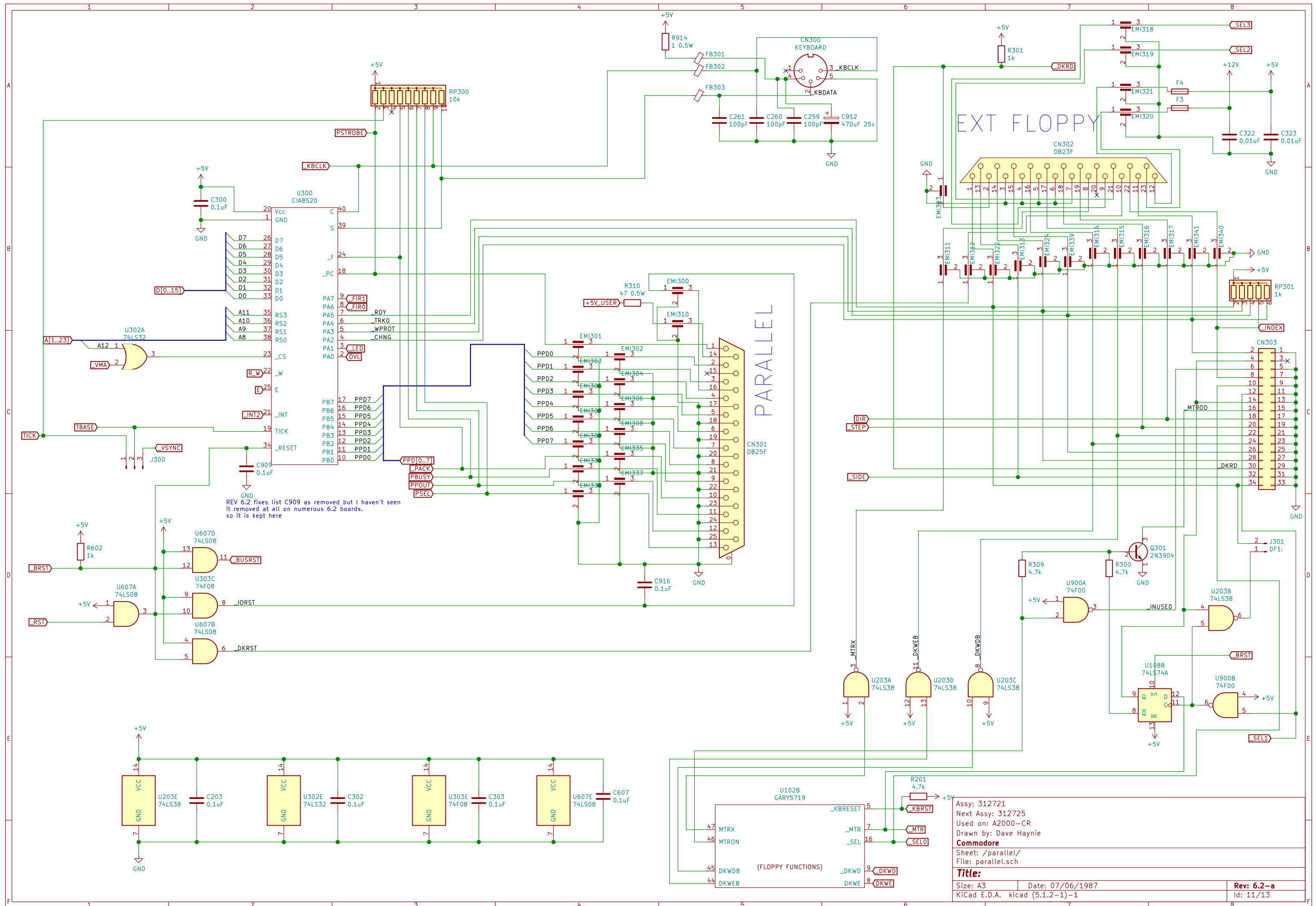
RP703
1k

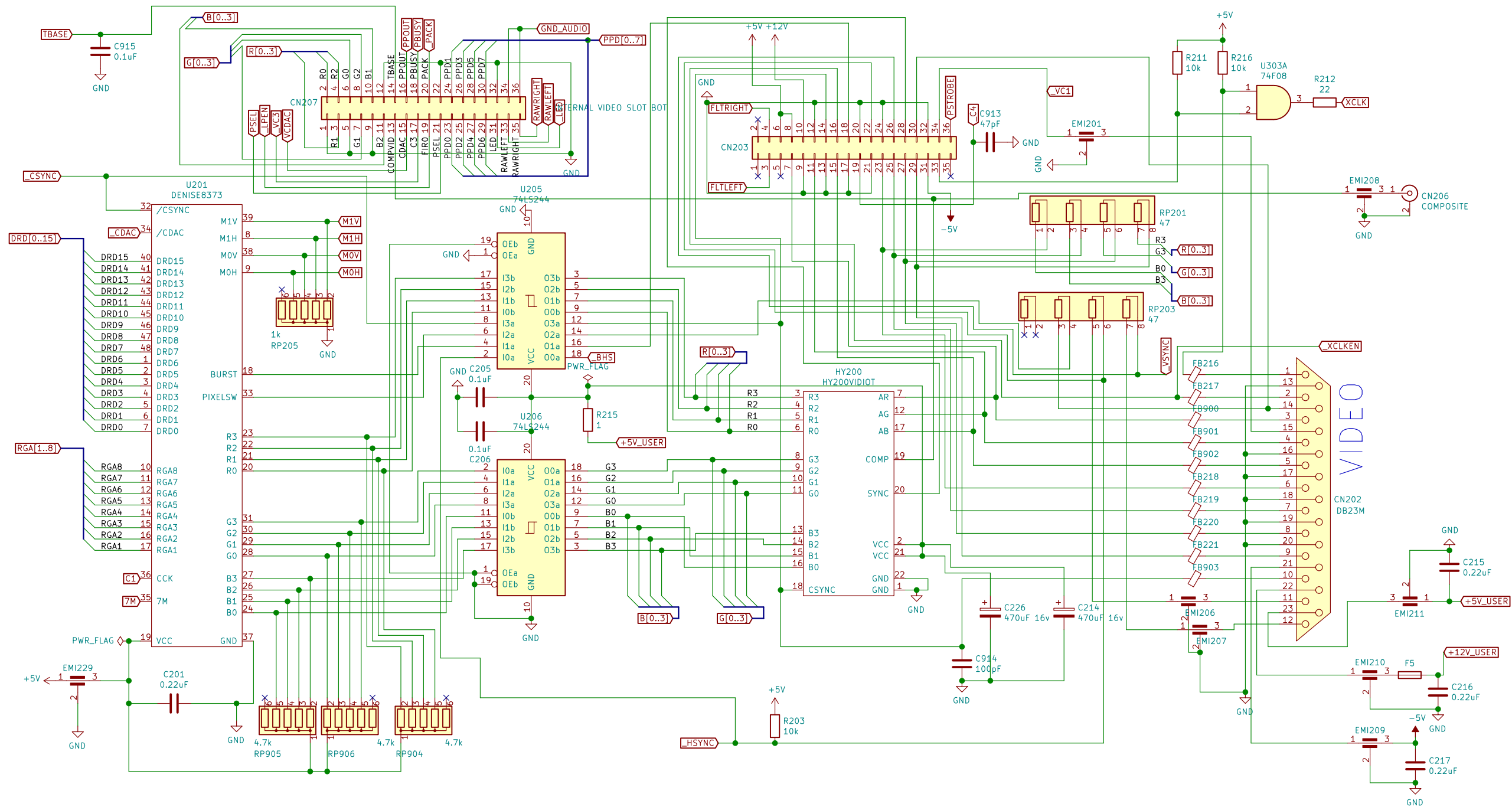
RP704
1k

RP705
4.7k

Note: R6 PCB - C789 is noted C788 and connected to _MASTER instead of D15







Assy: 312721		
Next Assy: 312725		
Used on: A2000-CR		
Drawn by: Dave Haynie		
Commodore		
Sheet: /denise/		
File: denise.sch		
Title:		
Size: A3	Date: 07/06/1987	Rev: 6.2-a
KiCad E.D.A. kicad (5.1.2-1)-1		Id: 12/13

