Date ____________ *Portfolio For (Name)* ________________________ *Page* ___

Due Date: End of class the day after the date at the bottom of this file
Peer Assessor's Name: __________________

Mark: _____ / 100 (Application)

AutoDesk Inventor for Problem Solving: Wooden Gears (2)

DO THE WORK DESCRIBED AND PUT THIS INPUT FILE IN YOUR ELECTRONIC PORTFOLIO

CONTENTS

1 Design Efficiency (Refer to the Previous File)

While the strategy in the assignment in the previous document (file 14_Inventor_2014_Gears.doc) seems to work ok, it may not be very efficient from the design perspective. How so?
Elements for a sample response are shown in blue font below.

1) Identify and discuss at least one "pro" ("good aspects") of the above design strategy for a pair of gears in a gear train.
   a) The recess into which a tooth will mesh is circular. So these can be cut using a drill bit, following a pattern around a circle. A jig can be set up on a drill press table extension to keep the gear in a fixed relationship with the drill bit tip, rotating the gear using an indexing arrangement to drill out the next recess.
   b) Note: We could perhaps set the tooth to be slightly smaller than the recess so there would be little to no binding as the teeth mesh together in a pair of mated gears.

2) Identify and discuss at least two "cons" ("bad aspects") of the above design strategy for a pair of gears in a gear train. Now consider a larger gear train of several more gears – are there any more "cons"?
   a) There is no relationship between number of teeth and the diameter of the gear plate, when, clearly there could be.
   b) There is no relationship between the diameter of a tooth and the diameter of the gear, when there could be.
   c) So it will be time consuming to generate another variant – ie another size, simply because you have to re-model the entire gear again.

d) In the *.iam file there is no clearly stated relationship between gear diameter and the distance between the centres of the gears, although we know what it is approximately. The two gear instances have just been butted up to each other without some kind of equation.
e) In a larger gear train, there will be multiple gears on a particular shaft and perhaps the gears even have to mesh back to a single shaft hosting another pair (or multiple) of gears.
f) There does not seem to be a clear way to make a simple parameter change which will be automatically propagated through a design to yield an improved variant, say a gear ratio that is closer to what we really want or need.
g) The circular tooth shape will not result in a uniform speed of rotation throughout the period of tooth surface contact. This will not be an issue in a clock gear train which only tick-tocs and moves when it is called upon to move.

3) Design a pair of meshing gears in a ratio of 2:1 using a flexible design methodology – you should be able to very easily generate gear pairs of various diameters. Hint: use a bit of basic circle geometry and Inventor's constraints. Drop off a short document (for items 1 and 2 above), at least one part file and one assembly file.
   a) See the strategy and solution in document 14a_Inventor_2014_Gears.doc for a 12:1 gear ratio – somewhat more complex than a 2:1 gear set because of the two-stage gearing that we used (4:1 and then 3:1 results in 12:1 overall).

2 An Overall Strategy for Wooden Gear Trains that is Modestly Flexible

2.1 *Requirements and Design Intent*

#	Requirement	Translate the Requirement Into Your Own Words (Confirm Your Understanding)	Design Intent –Design Goals to Address This Requirement
1	The clock must have a 12 hour clock face. For every full clockwise revolution of the minute hand, the hour hand must move 1/12 of a revolution clockwise. The minute and hour hands must move on the same circular face.	By the time the hour hand does one full revolution, the minute hand will go around 12 times. Hence the ratio of turns is 12:1. The minute and hour hands rotate independently on a common central axis.	To design a gear train with a 12:1 ratio and to simulate the relative motion of a minute hand and an hour hand of a traditional clock. We also don't want to have a huge gear that takes up a lot of physical space laterally. So the plan will be to use two pair of gears to obtain the 12:1 reduction in two stages. The central axis of the minute and hour hands will be chosen as the XZ axis in the model.

2.2 *Specifications – How We Will Deliver the Design Intent*

Note these Fundamentals:

- The teeth in gear pairs mesh precisely, tooth for tooth
- For a Gear1 having 32 teeth meshing with a Gear2 having 8 teeth
  - If Gear1 does one complete revolution, Gear2 will revolve 32/8 = 4 times
  - If Gear1 is turning clockwise, Gear2 will turn counter-clockwise

2.2.1 Minutes-Hours Gear Train

- Gear Ratios to be used
  - 4:1 (Large gear on and driving the hour shaft) (32:8 teeth)
  - 3:1 (Small gear driving the minutes shaft) (30:10 teeth)
- Let's put it another way…

Gear and Its Output Function	# of Teeth	Meshes (and on the same plane) with	# of Teeth	Gear Ratio
Gear1's shaft drives hour hand	32	Gear2	8	4:1 Gear2 turns 4 revs for 1 rev of Gear1
Gear4's shaft drives minute hand	10	Gear3	30	3:1 Gear4 turns 3 revs for 1 rev of Gear3
Gear4 revolves 12 times for each revolution of Gear1		Gear2 and Gear3 are both locked to a common shaft which is free to turn		4 x 3 =12
Gear1 and Gear4 are on concentric shafts rotating independently of one another on the XZ axis of the model				

2.3 Setting a Framework for Our Design Strategy – How Will We "Get There"?

2.3.1 Best Practices for Parametric Modelling and Problem-Solving – aka "Rubrics"

1) Be sure to refer to the following files in the pickup folder for marking scheme / criteria / "Rubrics":
   a) Inventor_Marking_Criteria.doc: Regarding CAD marks
   b) MarkingScheme_Probl_Solving_3DCAD.doc: Regarding Problem-Solving marks
   c) Rubric_Information_Processing: Regarding Thinking marks
   d) Written_Report_Rubric: Regarding Communication marks

Note: Other than the references to "Marks", there is nothing in the above "rubrics" that is specifically only for the classroom setting. These "Best Practices" are very "real-world" – to get you ready for your professional future in problem-solving!

2.4 Parameter-Naming Convention -- Suffixes for Model Parameters and User Parameters

1) The following convention of suffixes on parameter names will be used:
   a) _L suffix for a length
   b) _W suffix for a width
   c) _T suffix for a thickness
   d) _D suffix for a diameter

e) _ID suffix for inside diameter
f) _OD suffix for outside diameter
g) _R suffix for a radius
h) _O suffix for an offset from an edge
i) Parameters associated with a particular Gear, eg Gear1 will have prefix Gear1_
   i) Gear2_
   ii) Gear3_
   iii) Etc
j) A dimensional relationship between a pair of Gears will be named as follows, for example:
   i) Shaft1_2_Separ (this is the distance between the centres of the shafts of Gear1 and Gear2)

2.5 Some System Parameters (aka User Parameters)

The following system parameters will be used as the basis for setting values of other model parameters (eg Gear1_T=Gear_T)

1) All Gears will have the same thickness
   i) Gear_T = 1.90 cm (0.748", ie approx. 3/4")
2) All Teeth and their meshing Recesses (ie the hollow bit into which the tooth "fits" as the gears turn) have the same radius
   a) Recess_R=0.5 cm
3) All Shafts to which intermediate gears are locked (ie other than Second, Minute, and Hour shafts) have the same diameter
   a) Shaft_D=1.9 cm
4) Concentric Shafts for the time-display hands
   a) HrShaft_D=
   b) MinShaft_D=0.95 cm
   c) SecondShaft_D =0.5 cm
5) All Washers have the following details:
   a) ShaftWasher_ID= Shaft_D + 0.1 cm
   b) ShaftWasher_OD= ShaftWasher_ID + 1.5 cm
   c) ShaftWasher_T=0.2 cm
6) All Support Brackets (ie structural framework) have the same thickness
   a) SuppBrkt_T=1.90 cm

2.6 Multiple Solids Design Strategy -- Master.ipt Part File

The entire system will be modelled in 3D using the so-called "Middle-Out" or "Blended" Strategy, aka "Multiple Solids". The design work will be done in one "Master.ipt" file. All of the distinct solid parts will co-exist with one another in the master file. When extruding a feature, "New Solid" will be chosen as the extrusion type. "Views" of the design may be taken for various purposes – but the design work should all be done in the original "Master" file. For example, assemblies and shop drawings of various related parts may be produced for the purposes of Quality Assurance or shop fabrication.

2.6.1 Model Versioning / File Naming Convention

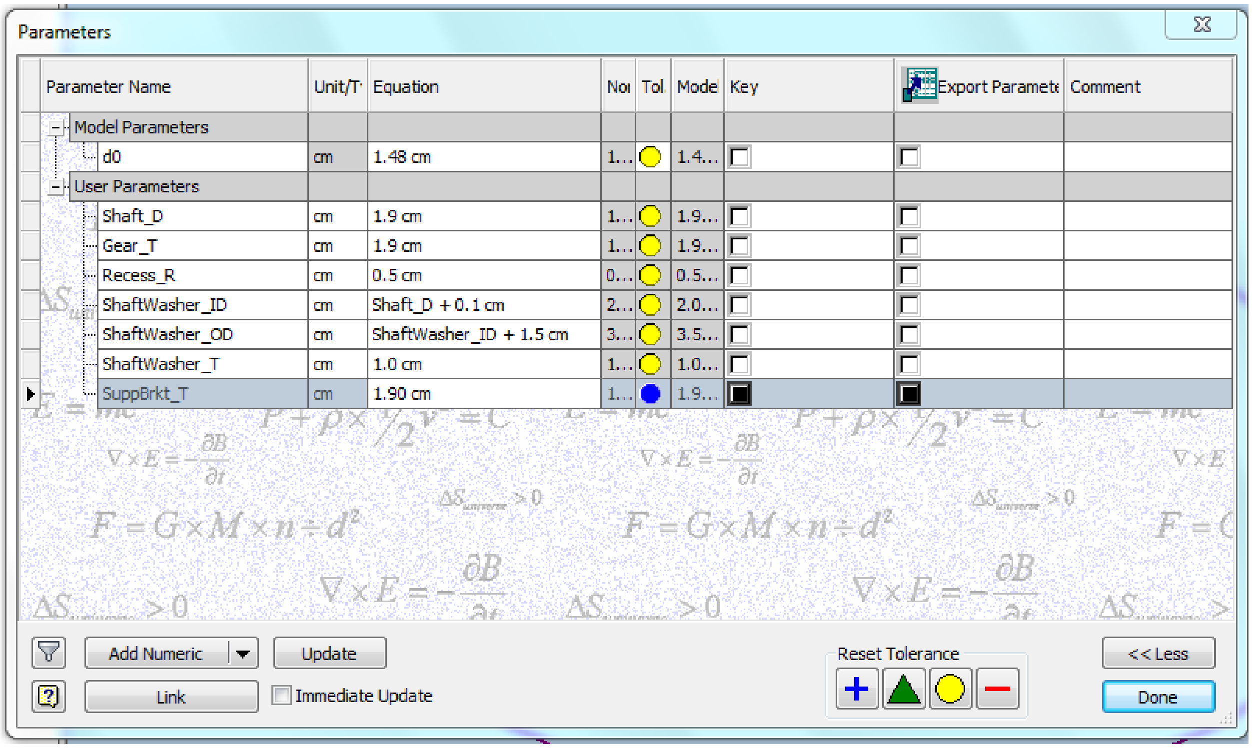
1) Save the part as YourName_ClockGearTrain_Master_current.ipt – all design work / modifications will be done in this file.
2) Always drop off the entire project folder – not just the master.ipt file
3) Immediately after you drop off version 1 of your project folder (and whatever individual ipt, iam and idw files that go with it):
   a) Make a copy of the entire project folder – give the folder the suffix V1_monthddyyyy
      i) Store this version 1 folder in a "PastVersions" folder under your ClockGearTrain project folder
      ii) Rename the Master.ipt as Master_V1_monthddyyyy (ie delete the word "current").
      iii) This previous version will be available if you ever need to go back to it for some reason.
4) Continue all design work in YourName_ClockGearTrain_Master_current.ipt.
   a) Changes will be reflected in individual ipt and idw and iam files
   b) Do not make any dimension changes in idw or in iam files
   c) Do not make any dimension changes in part mode (3D) of the master.ipt
   d) To change a dimension, always go to the sketch in which the dimension was created
      i) The sketch should already be fully constrained – make sure there are no degrees of freedom before changing a dimension
      ii) After changing the dimension, make sure that the sketch updates correctly
         (1) You may need to click Update / Update Local
         (2) Make sure there are no degrees of freedom

2.6.2 Drive1 (4:1 Gear Ratio)

Note:
The strategy to generate a gear variant (eg change the number of teeth as shown below) involves some modest additional work. You are encouraged to experiment to discover how you can avoid this fairly minor issue.

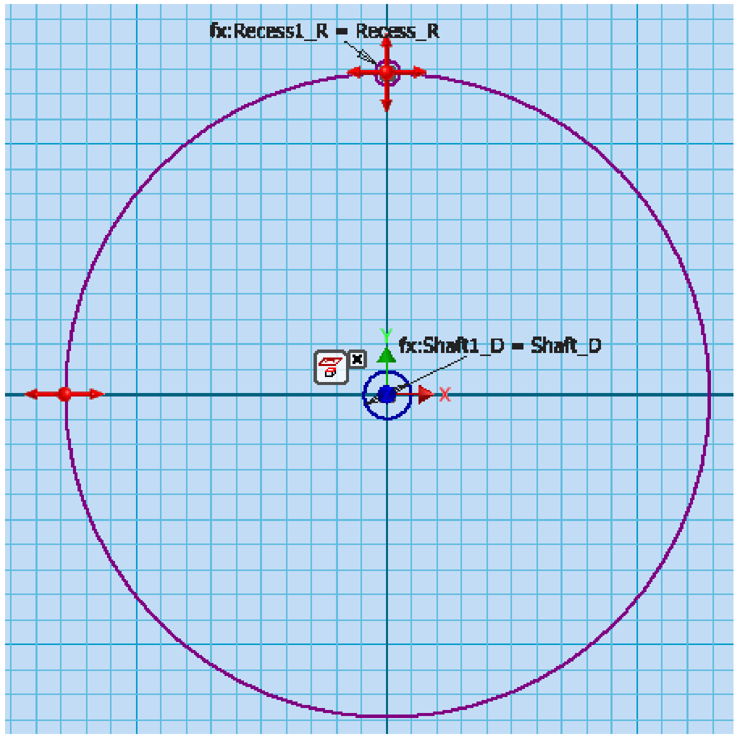
1) Design strategy decision
   a) I will create a Master.ipt part file, working in a "Middle-Out" fashion so that I can generate variants by changing the values of several parameters.
   b) Application / Manage / Project / New Project / Single User
      i) Start a new Project called YourName_ClockGearTrain in folder
   c) Application / New / NewFromTemplate
      i) Start a new part file using your PromptConstraint_10cmMajor_template.ipt
         (1) Your new ClockGearTrain.ipj should be the default project file when you "Create" the part file
2) **Gear1 (32 teeth)** -- Sketch1 – On the XY Plane
   a) Centre a circle at the origin of the XY plane as an approximate outside diameter of a gear – do not dimension yet
   b) Centre a small circle at the origin of the XY plane – dimension as Shaft1_D=1.9 cm (Right click and choose Diameter as the Dimension Type)
   c) Click Finish sketch to save the part file (certainly not finished yet!)
3) But first set up these User Parameters
   a) Manage / Parameters
      i) Click Add Numeric in the bottom left corner of the dialog

ii) Enter the User Parameters shown below – enter into both the Name and Equation columns

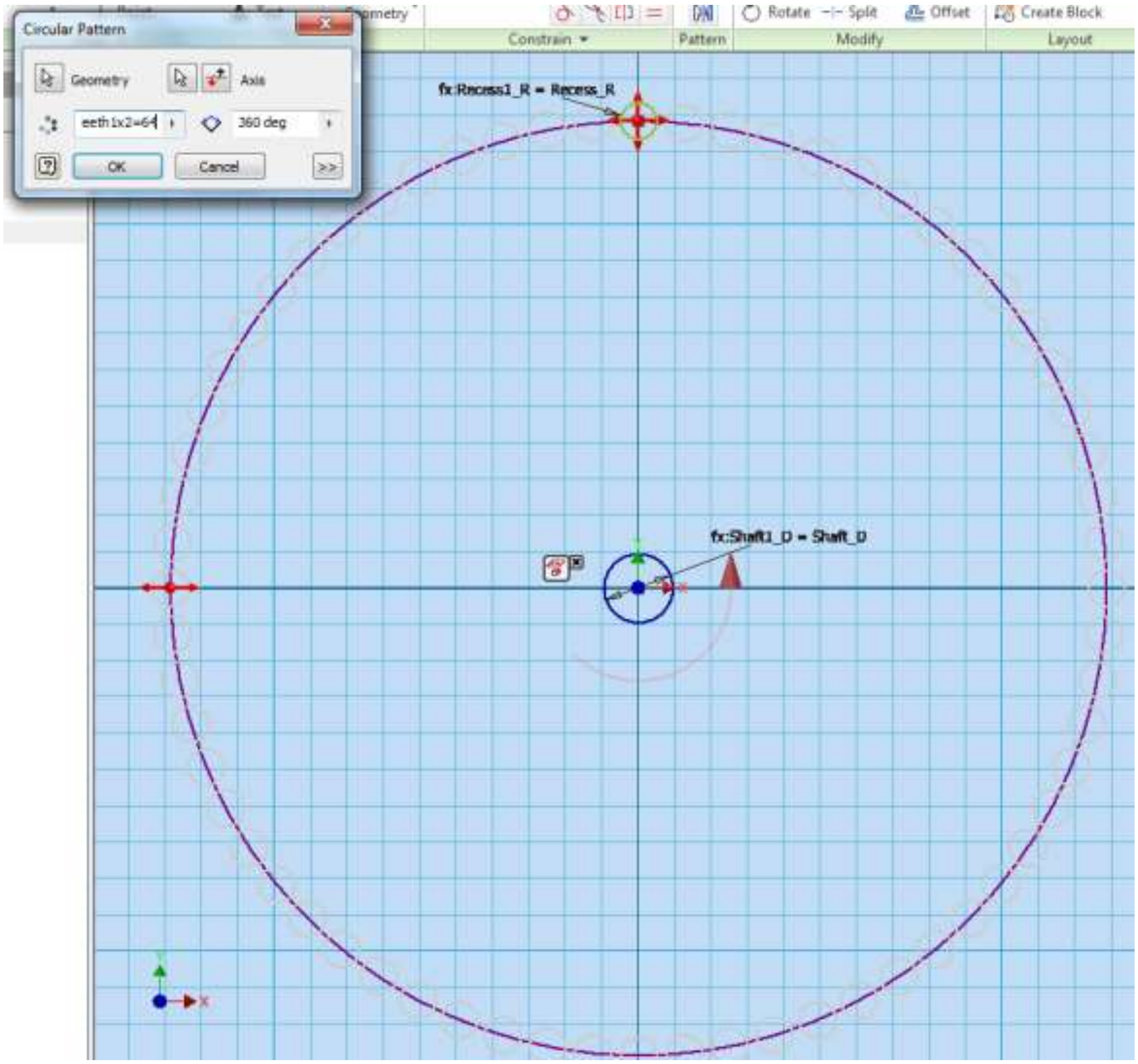


4) Save the. ipt Part file using the name YourName_ClockGearTrain_Master_current.ipt

5) In the Model Browser, right click on Sketch1: Edit Sketch
   a) Manage / Parameters
      i) Change paramter Shaft1_D=1.9 cm to Shaft1_D=Shaft_D
      ii) Update / Local Update
   b) Centre a small circle on the Y axis very close to top of large circle – apply coincident constraint on centre of small circle and the large circle circumference on Y axis
      i) Dimension this circle: Recess1_R=Recess_R (where dimension type = radius)
      ii) Drag the large circle such that you can "ball-park" about 16 of the Recess1_R circles within each quadrant of the large circle. Note the degrees of freedom in the sketch below.

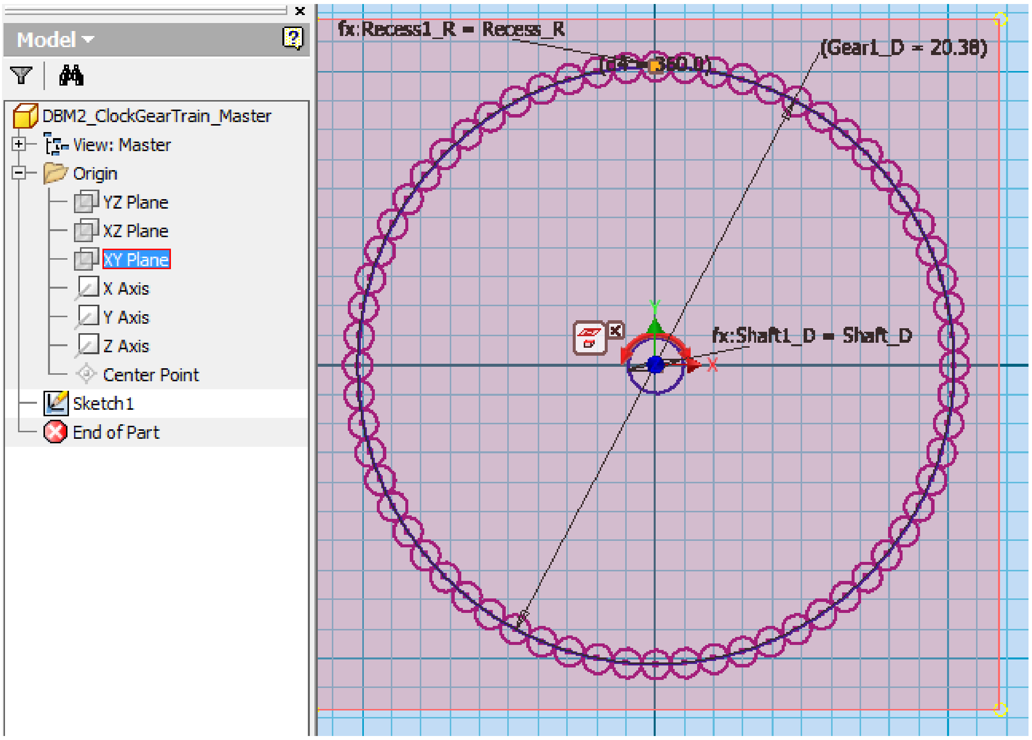


iii) In Pattern Panel
   (1) Circular Pattern:
      (a) Select Recess1_R circle, Enter
      (b) Enter number of instances of the circle NumTeeth1x2=64 (Note: the actual number of teeth on the gear circumference will be 64/2=32)
      (c) Use the large circle as the path axis for a full 360 revolution

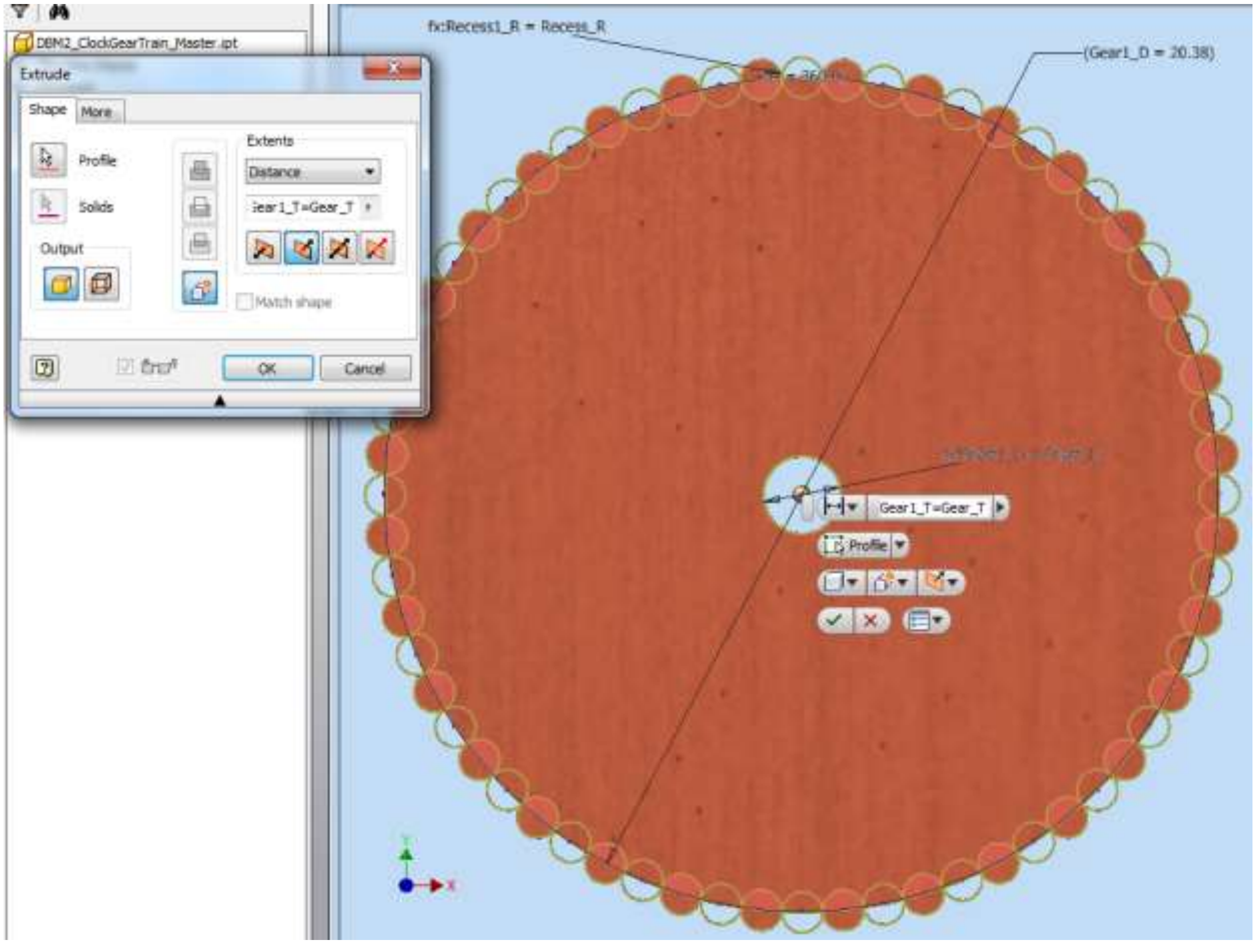


iv) Tangent constraint: select two adjacent small circles on the large circle's circumference
   (1) The large circle will adjust in diameter too

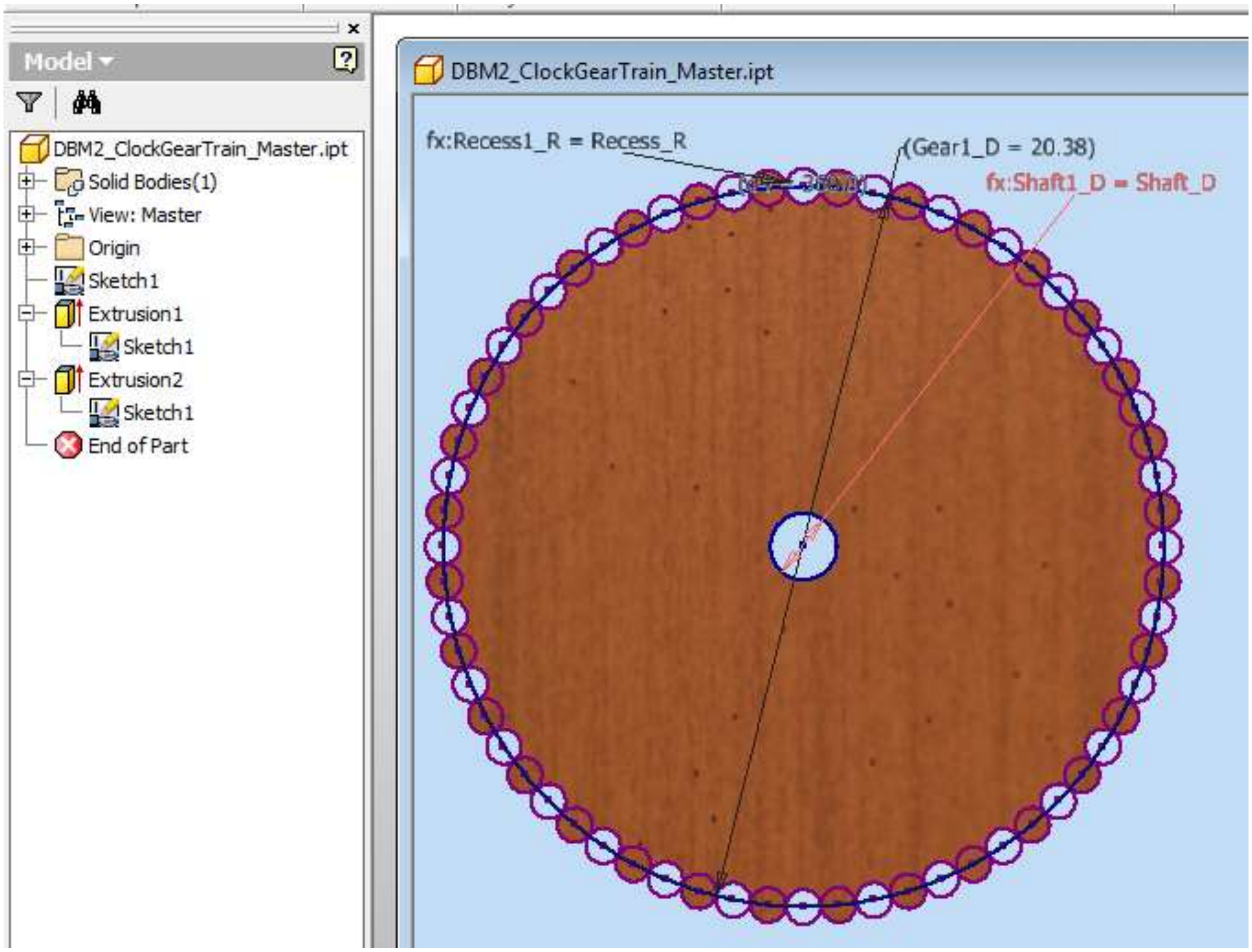
v) Try to dimension the diameter of the large circle – a driven dimension will result – accept the driven dimension using the name Gear1_D (in the Parameters table – Reference section)



c) Finish sketch
d) Save the part (YourName_ClockGearTrain_Master_current.ipt) – all design work / modifications will be done in this file
e) Extrusions – 3D Model Ribbon
   i) Into the screen, extrude the large circle's profile **and** every other small (Recess1_R) circle (ie 32 of them) – Thickness of the extrusion is Gear1_T=Gear_T



i) Under Extrusion1 in the Model Browser, right click on Sketch1: Share sketch
   (1) Into the screen, Extrude-Cut every second Recess1_R circle – RecessCut1_T=Gear_T

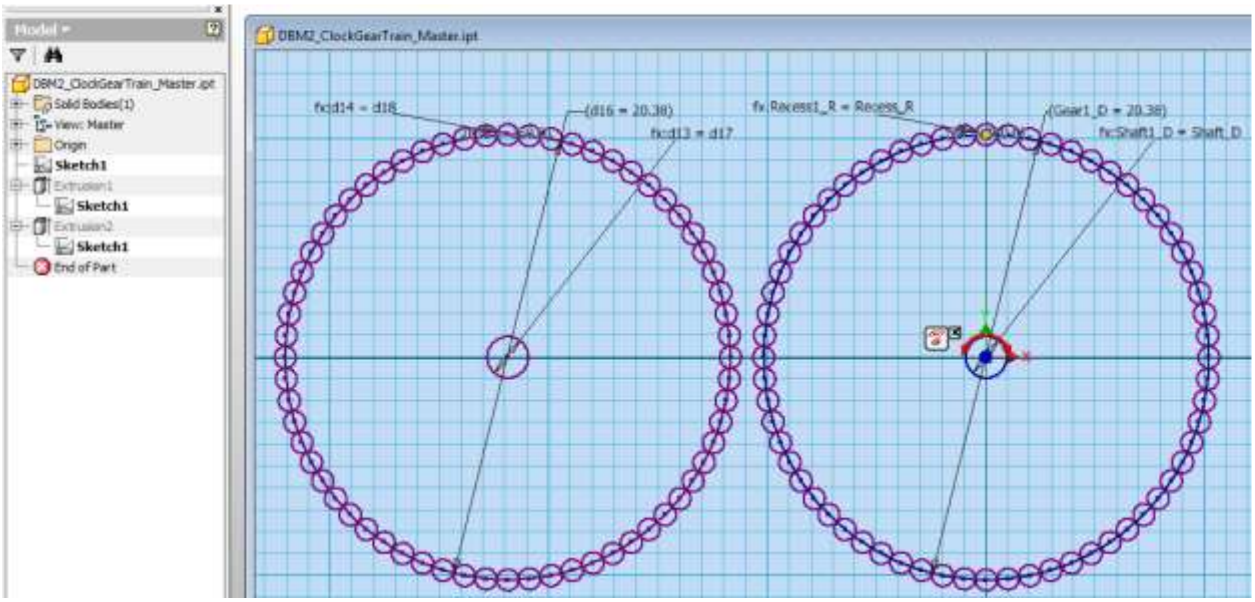


   b) Save part file

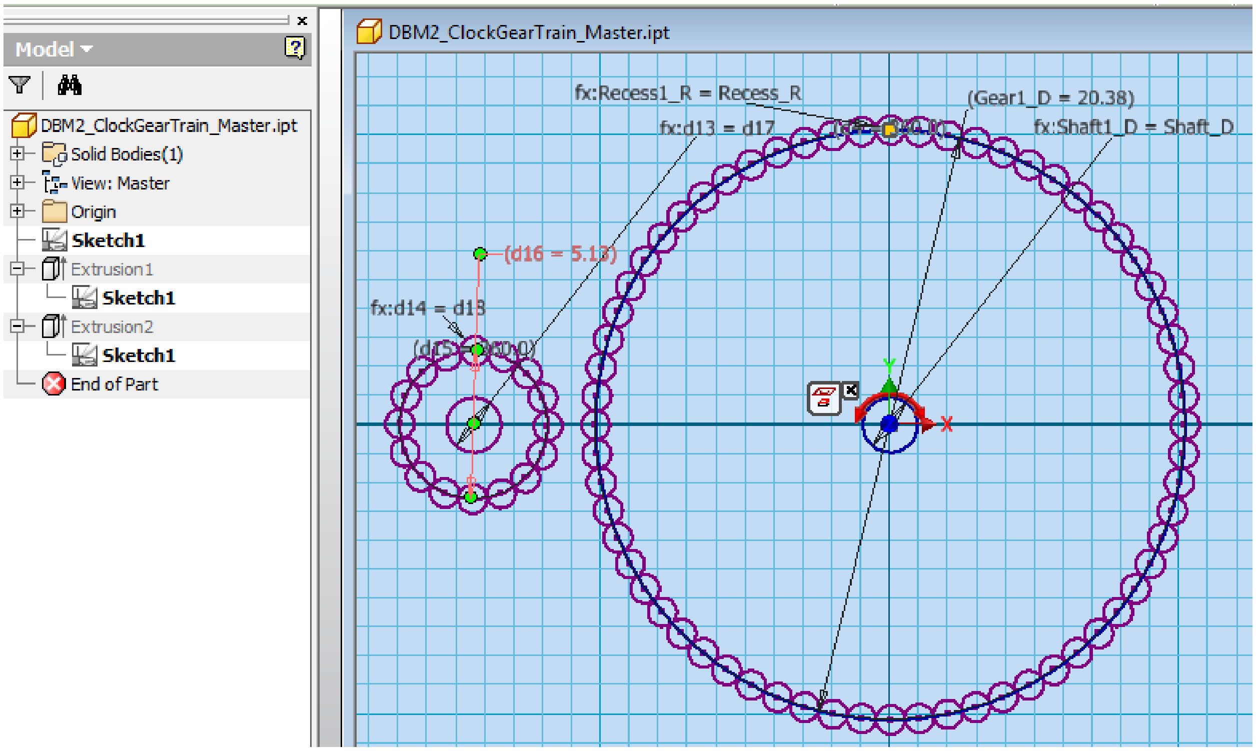
2) Note the following significant mathematical issues and relations regarding this first gear (Gear1):
   a) There are 32 Teeth in Gear1
   b) Tooth size is based on a nominal 1.0 cm dia drill bit
   c) Circumference of large circle is π*Gear1_D = 3.14 x 20.38 = 63.993
   d) Circumference of large circle can be approximated by (NumTeeth1x2)(Recess1_R)2 =64*1=64.00
   e) Shaft1_O – ie the distance from the centre of shaft to centre of tooth is half of Gear1_D = 10.19
   f) Shaft1_O can be approximated by (NumTeeth1x2)(Recess1_R)(2) / 3.14/2=64(0.5)/3.14 =10.191
   g) For a gear having at least 32 teeth, there is only a tiny difference in the two Shaft Offset calculations.
      i) DO A BIT OF MATH: Satisfy yourself that even a gear with as few as eight 1.0 cm diameter teeth will show a negligible difference between the two calculations for the Shaft offset
3) Now define the Shaft offset for Gear1 as a new User Parameter (it will appear at the bottom of the Parameters table)
   a) Shaft1_O = NumTeeth1x2 * Recess1_R / 3.14 ul

4) Gear2 (8 teeth)

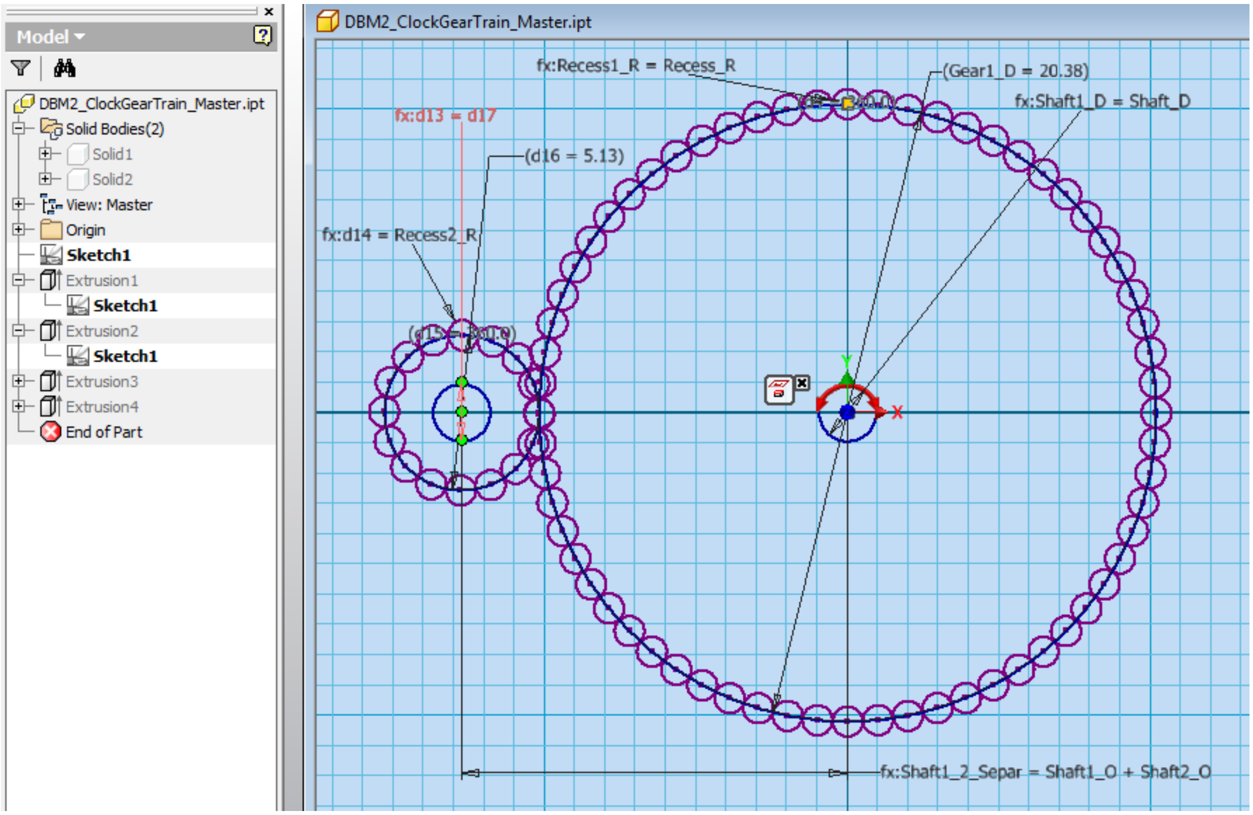
5) Right click on Sketch1: Edit Sketch
   a) Use Modify / Copy to place a copy of the 32 tooth Gear1 to the left of it on the screen where the centre of its shaft is also on the X axis
   b) Use our parameter naming convention to rename parameters that are automatically created for Gear2



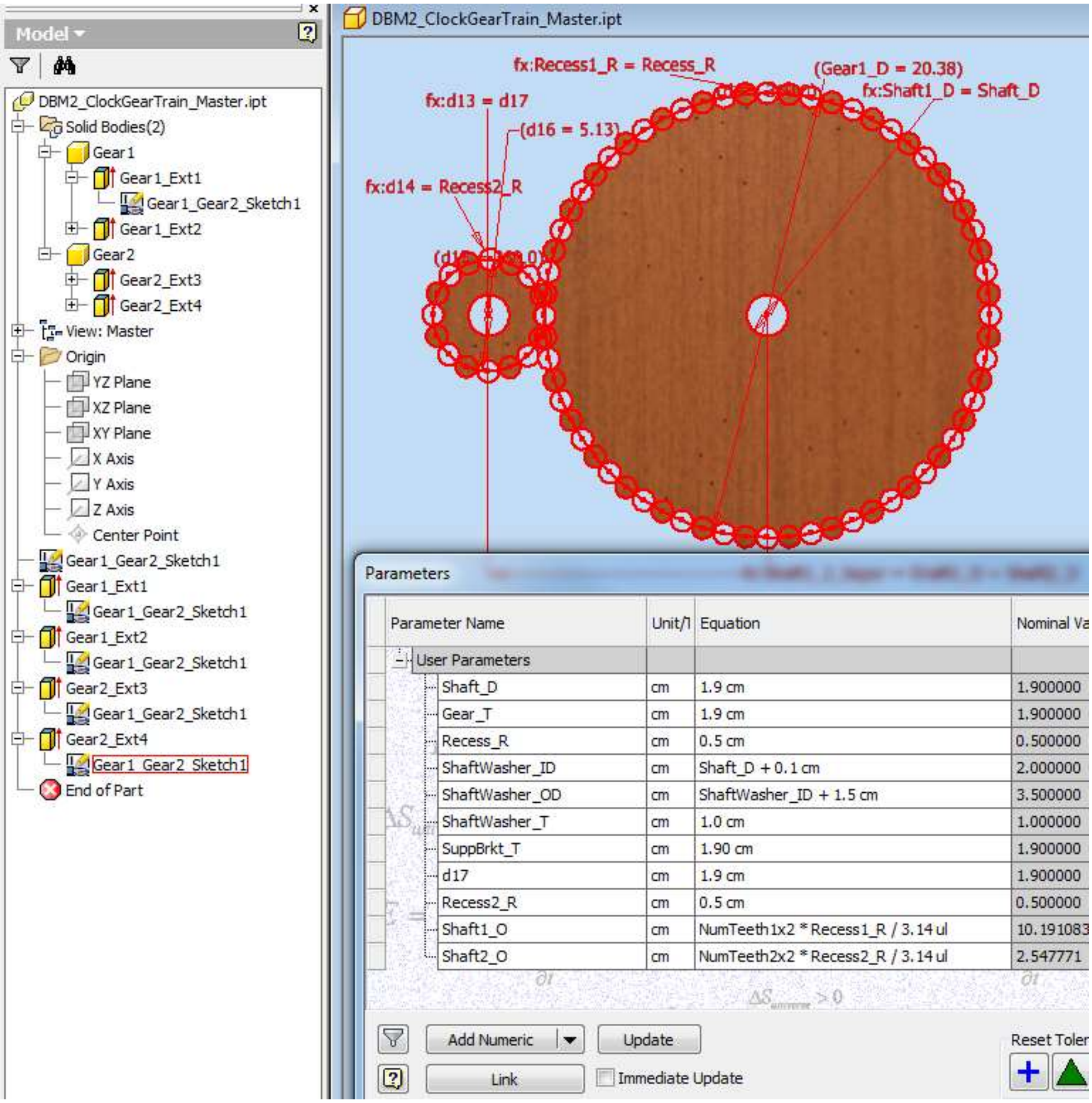
6) This second gear (on the left) will be called Gear2. On this Gear2, do this:
   a) Click on one of the circles representing a tooth; right click; edit pattern
      i) NumTeeth2x2 = NumTeeth1x2 / 4
      ii) This will give a gear with 8 teeth (the Gear1 to Gear2 ratio will be 4:1)
   b) Apply the Tangent constraint to the edges of two of the teeth circles
   c) Ensure that the centre of the shaft of Gear2 is constrained to the X axis – try to drag upward or downward
   d) If you had not properly constrained the shaft to the X-axis, do this:
      i) Use horizontal constraint to lock the centre of Gear2 to the X axis – ie use the centres of the 2 gears with the horizontal constraint



e) Finish sketch
f) To Extrude Gear2 – BE SURE TO USE NEW SOLID and extrude into the screen
  i) **New Solid** made up of the gear plate and the 8 teeth: Gear2_T=Gear_T
  ii) **Cut** the 8 alternate Recess_R (cut through the gear plate solid): Recess2_T=Gear_T
    (1) Select Solid2 from the list of 2 solids in the Model browser (tells application which solids it must cut from)
  iii) Rename default parameters according to our convention
g) New User parameter: Shaft2_O = NumTeeth2x2 * Recess2_R / 3.14 ul
h) Edit Sketch1: Dimension the separation between the two gear centres
  i) Shaft1_2_Separ=Shaft1_O+Shaft2_O
  ii) The two gears are now the correct distance apart
  iii) But the two gears may not mesh properly at this stage – we will fix this in the assembly file



   i) Then finish sketch
7) Rename Solids meaningfully
   a) Change Solid1 to Gear1
   b) Change Solid2 to Gear2
8) Rename Extrusions and Sketches as shown below. This may seem like extra work, but it is well-worth it in a large design project.



3 Inputs / Knowledge / Understanding I Still Need For This Module

Give each issue a number for future reference:

NOTE: In the feedback, the Peer Assessor must “make the student think” – not give the student the answer! Be sure to include comments justifying the mark that you are giving.

Assessor’s Name and Additional Notes: