13 Easy Step by Step Tutorial to Model Involute Spur Gear in IV 9

by : Taufik Haryanto aka Solus Emsu
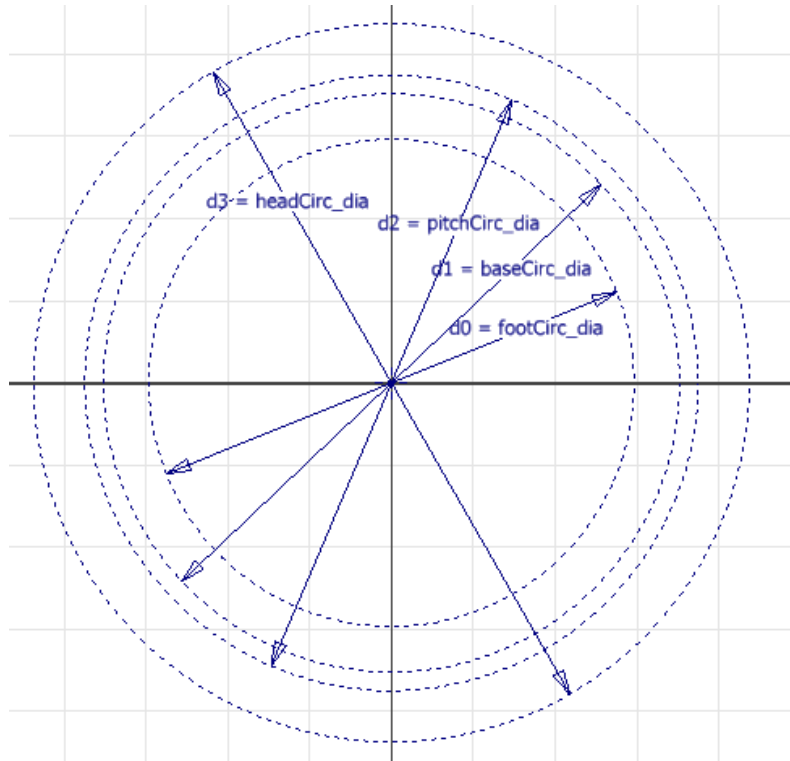
Mechanical Eng. Dept. of UGM

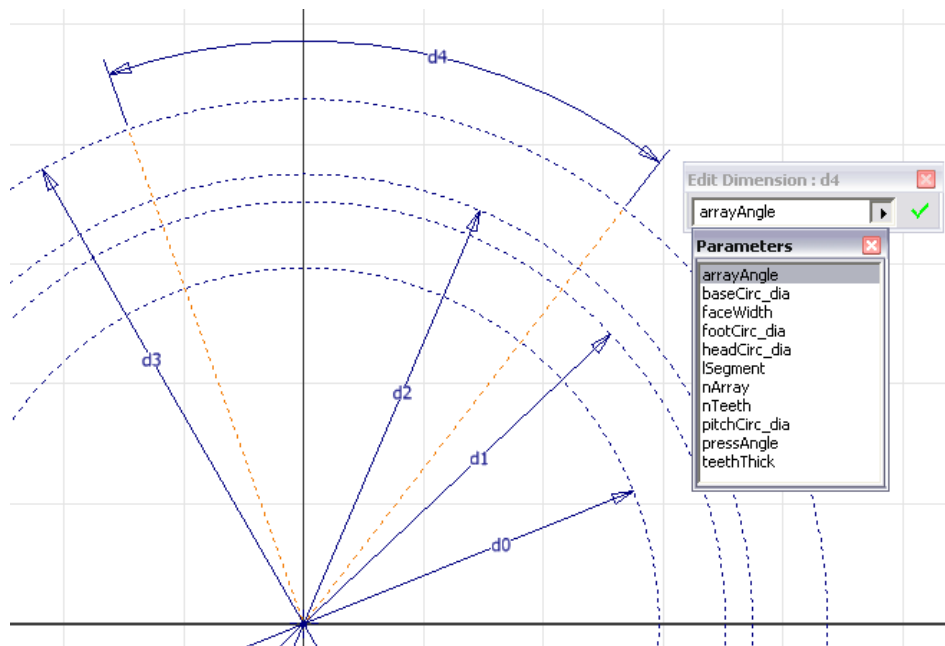|	A	B	C	D
1	Parameters	Magnitude	Unit	comments
2	footCirc_dia	11,875	mm	foot/dedendum circle diameter
3	baseCirc_dia	14,095	mm	base circle dia.
4	pitchCirc_dia	15,000	mm	pitch circle dia.
5	headCirc_dia	17,500	mm	head/addendum circle dia.
6	pressAngle	20,000	deg	pressure angle
7	nTeeth	12,000	ul	number of teeth
8	teethThick	1,964	mm	teeth thickness
9	faceWidth	6,000	mm	width of face
10	arrayAngle	45,000	deg	angle of array>>arbitrary
11	nArray	15,000	ul	number of array>>arbitrary
12	lSegment	0,396	mm	length of arc per array segment
13				
14			black =	from design calculations
15			pink =	your decission
16			yellow hi-lited =	calculated with formula

1.) First of all let's create Excell table contained parameters that'll define the Spur Gear profile. The black colored params referred to params that is acquired by design calcs (from your Mech. Hand books such as Newman, Kiyokatsu Suga etc… I'm using Kiyokatsu Suga). The pink params is decided by your choice. The "arrayAngle" is an angle between the first and the last array line (not the angle measured between one line and the next one). I used 45 deg for arrayAngle and 15 for array numbers. The yellow highlighted param is calculated with formula you can see on the pic below (I used 3 decimal places). Then linked em to the IV file that we've been working.

| SUM	▼	✕ ✓ =	=ROUNDUP(((PI()*B3*B10)/(360*(B11-1)));3)

📄 Involute Spurs Gears Profile.xls

	A	B	C	D
1	Parameters	Magnitude	Unit	comments
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12	lSegment	=ROUND	mm	length of arc per array segment
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14			black =	from design calculations
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d3 = headCirc_dia
d2 = pitchCirc_dia
d1 = baseCirc_dia
d0 = footCirc_dia

2.) Now, going to IV… I always create point first right in the center of coord and fix it. Then, create 4 construction circles, and dimension them refer to the pic above (concident em with the fixed point we first created).


Edit Dimension : d4
arrayAngle

Parameters
arrayAngle
baseCirc_dia
faceWidth
footCirc_dia
headCirc_dia
lSegment
nArray
nTeeth
pitchCirc_dia
pressAngle
teethThick

3.) Add 2 construction straight lines and constrain em with arrayAngle parameter dimension (d4 on the pic above)

Circular Pattern

Geometry   Axis

nArray   arrayAngle

OK   Cancel   >>
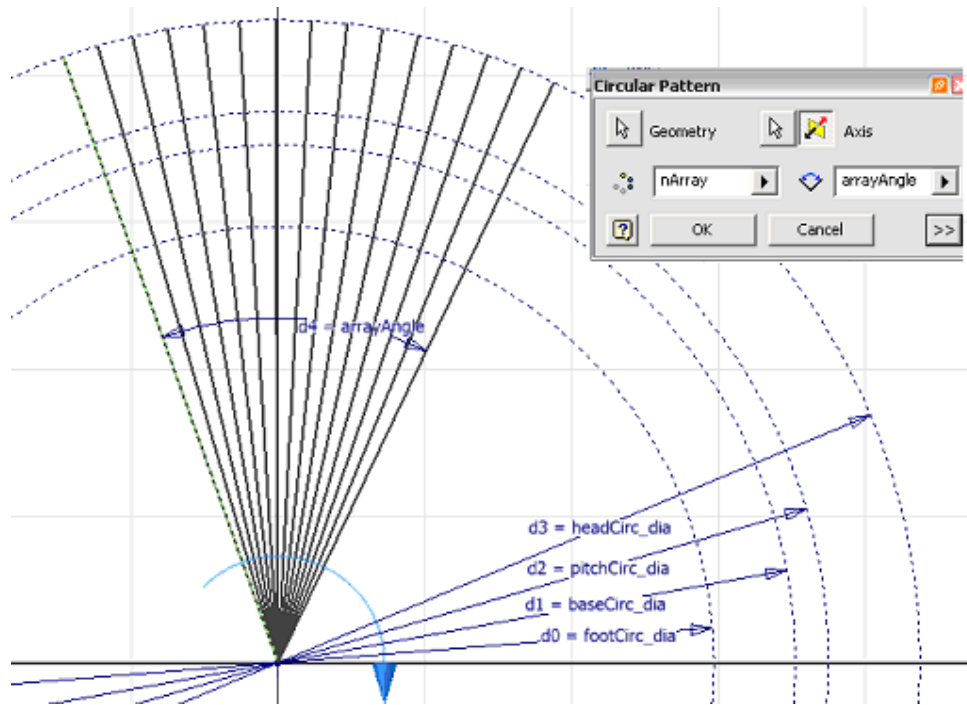
3.) Now make a circular pattern from the 1st constru line. Fill the dialog refer to the pic above. Then erase the second constru line that we've made before. If you properly did this, it should seems like the pic below. Note that the pattern is using constru line, **and the d4 dim is erased**. Someone maybe will ask me why I made d4 dim then…? Well, it's just to show you that the pattern we being created is correct.
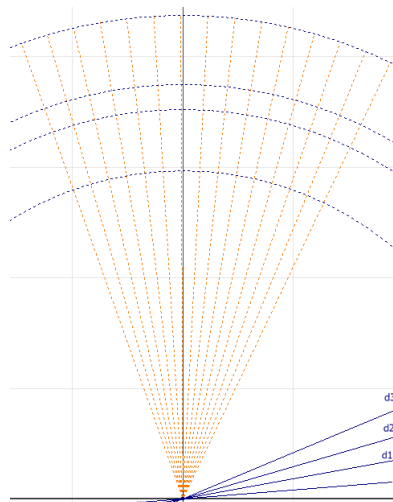

4.) Now create a line, tangensial to base circle, coincident with base circle and second arrayed line.
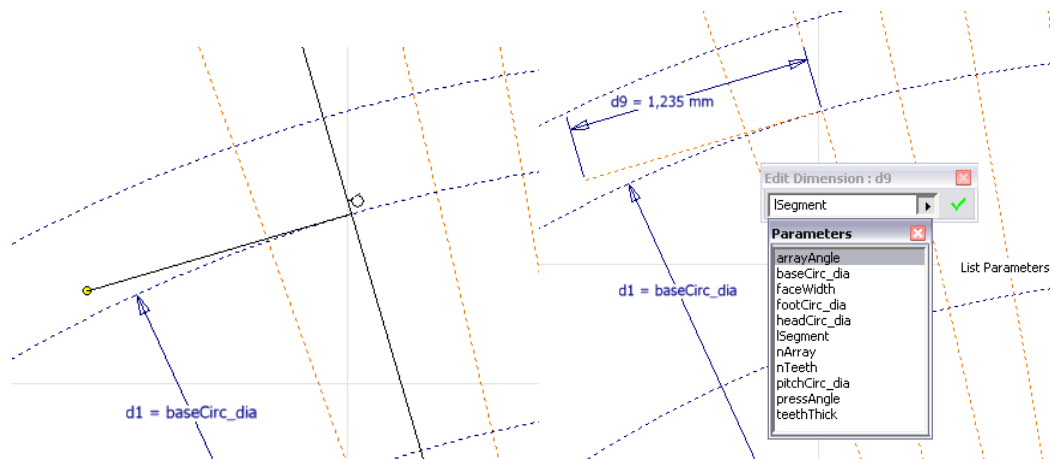
Dimension it with **lSegment** (don't forget to align the dim). Now create the second line, it should coincident with the third arrayed line and base circle, and tangensial with base circle. Dimension it with **2*lSegment.** Create the third line like 2 lines before… it coincident with base circle and 3rd arrayed line and tangensial with base circle… and so son. Dimension the 3rd line with **3*lSegment** , the 4th with **4*lSegment** …and the nth with **n*lSegment (do not forget to align the dims** or you can set the dim preference to set the align dim as default). Now it should be like this pic below…
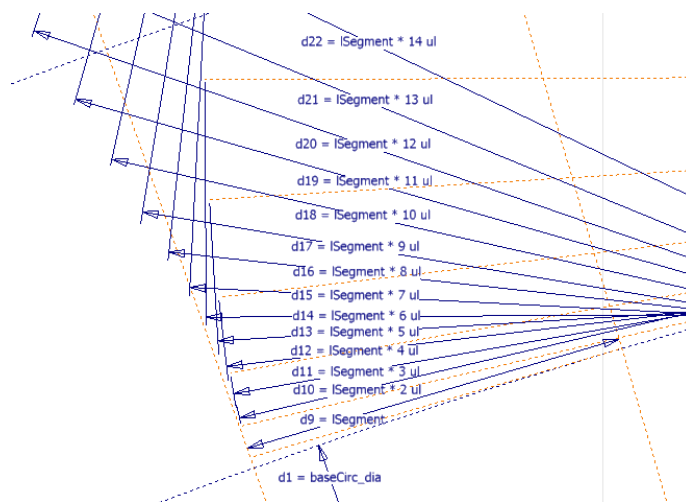

5.) Now turn off the construction button. Then create spline and join the edge point of "lSegment*n" line start from cross point of 1st arrayed line and base circle, and end it at the edge point of **lSegment*14** line. Then cut the spline on the line between edge point of **lSegment*14** line and head circle.
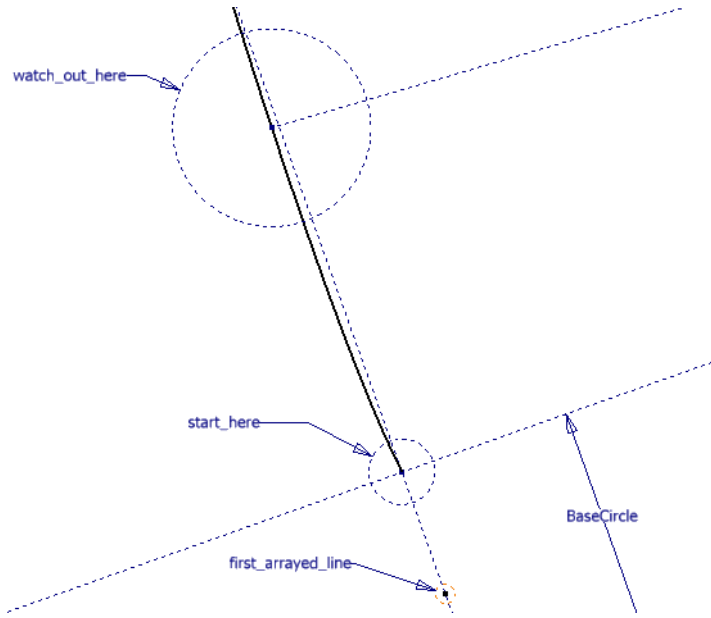
6.) Now we have involute profile… it should be look like this pic below. To fix em, dimension the 1st and 15th arrayed line with param **arrayAngle** (d24 on the pic below), then dimension the 1st (or the last line) with an arbitrary angle as you like (d23 on the pic below).
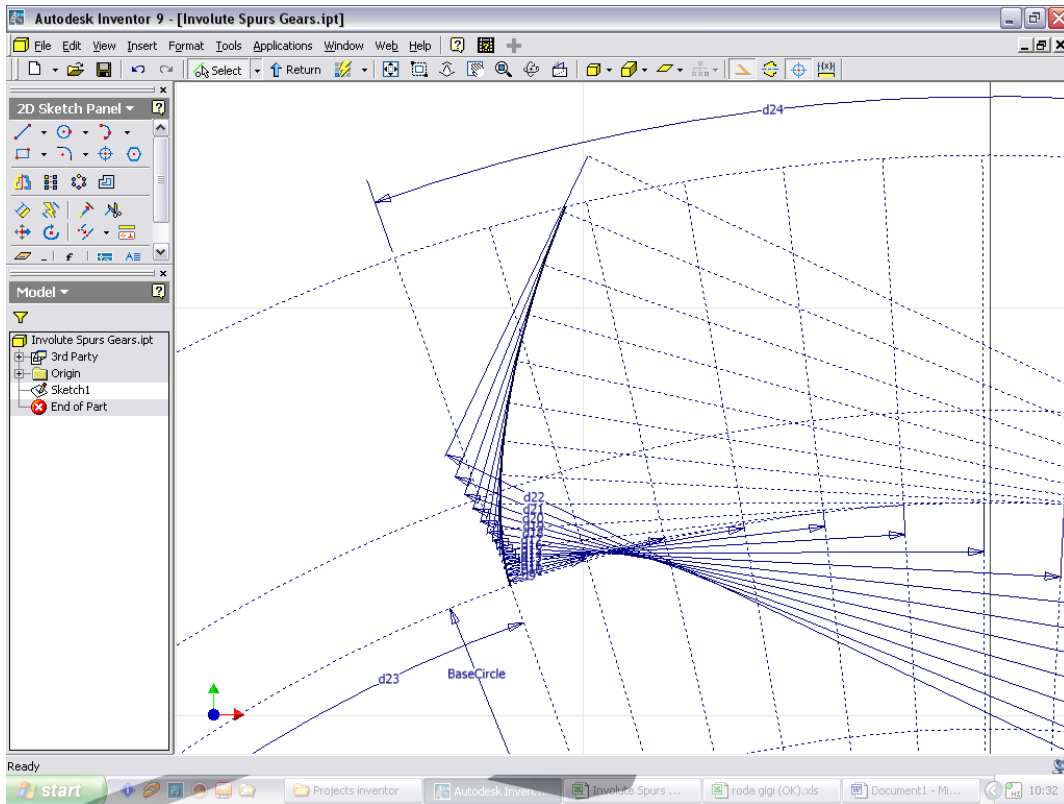
7.) Now add the tooth thickness. Create constru line starting from the cross point of **Pitch Circle** and the spline that we've created. Constrain the end edge point of the line with coincident constrain to the other side of pitch circle (far enough from starting point), but watch out to not constrain it with cross point of other lines. Add two other lines, starting from the center point of circles and end it at the two edge point of the first line we've created. It should be like this then (notice the yellow and red lines).
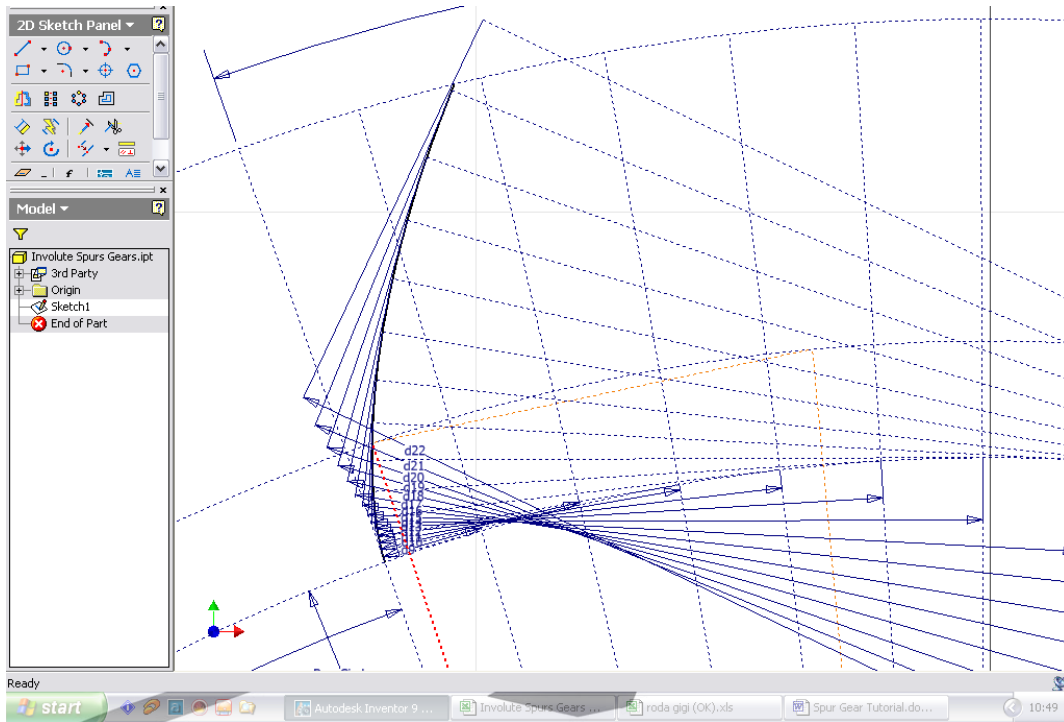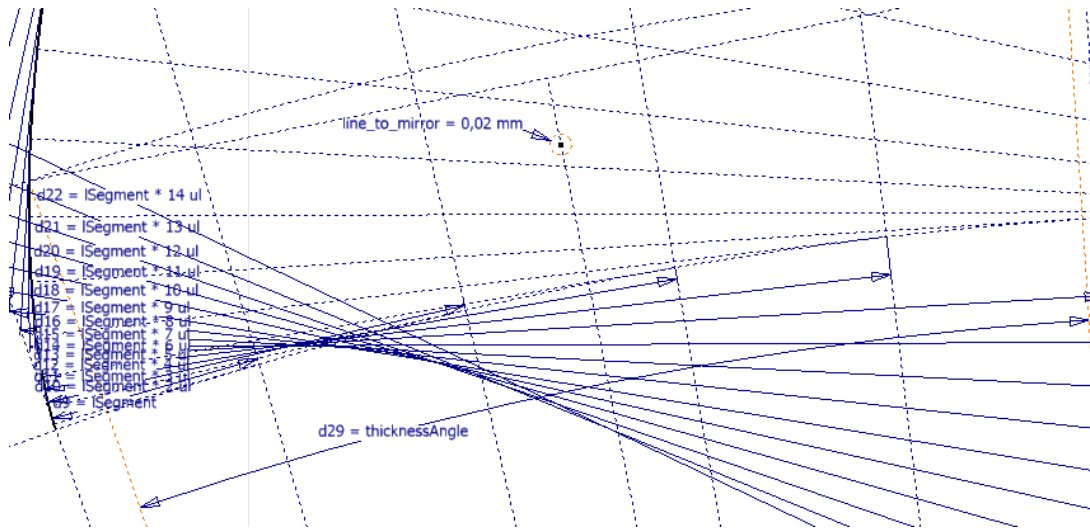

Open the param dialogue and add new user param. Let's name it **thicknessAngle**, with unit degree (deg). Fill the equation cell with this formula

$$(teethThick*360 \ deg)/((22/7)*pitchCirc\_dia)$$

it's just a formula to converse the teeth thickness to an equal angle (I'm sorry I can't say this properly in English)… Remember to type **deg** after **360**, it doesn't matter if you don't type the other unit.

Now going back to sketch env. Constrain the two constru straight lines with angle dimension, fill the dim cell with **thicknessAngle** param.

Add other constru line, starting form circles' center point and end it at the middle point of "bow string" line. This line will be used to mirror the involute profil. Now… mirror it, whatelse?

line_to_mirror = 0,02 mm

d22 = lSegment * 14 ul
d21 = lSegment * 13 ul
d20 = lSegment * 12 ul
d19 = lSegment * 11 ul
d18 = lSegment * 10 ul
d17 = lSegment * 9 ul
d16 = lSegment * 8 ul
d15 = lSegment * 7 ul
d14 = lSegment * 6 ul
d13 = lSegment * 5 ul
d12 = lSegment * 4 ul
d11 = lSegment * 3 ul
d10 = lSegment * 2 ul
d9 = lSegment

d29 = thicknessAngle

8.) Now let's add arc (whatever the methods) from this point to this point… (see pic below).

to_this_point

add_arc_from_this_point

d22
d21
d20
d19
d18
d17
d16
d15
d14
d13
d12

d29
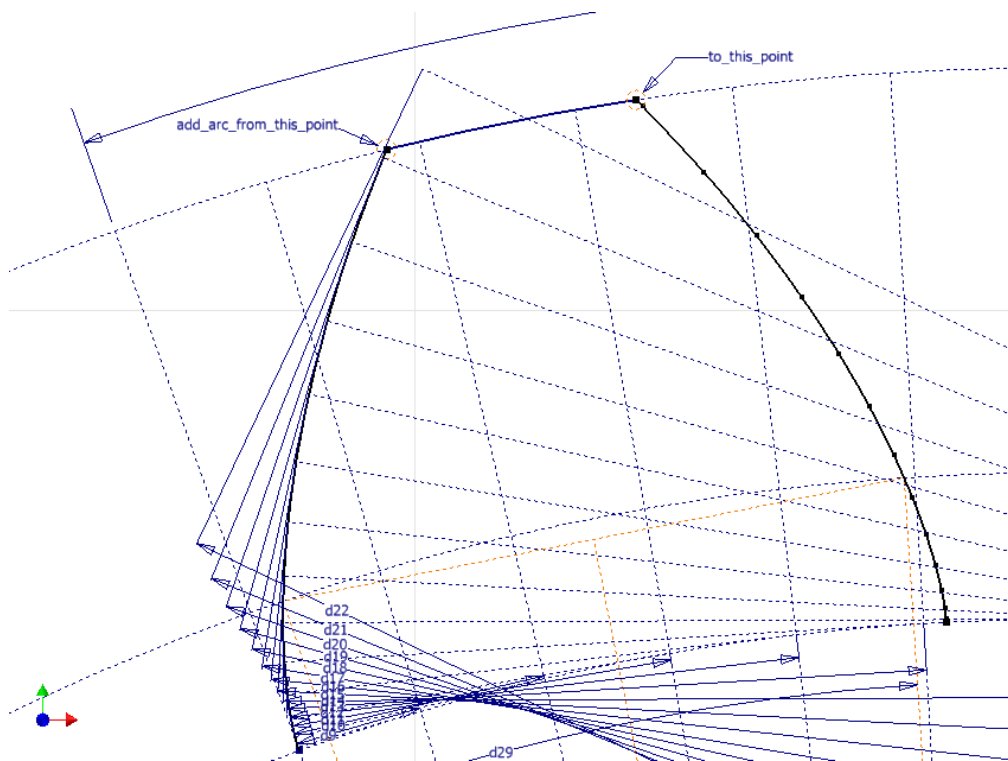
9.) Then… create line starting from the root point of involute profil on the left side, to the cross point of 1st arrayed line and base circle. Then from that point create arc to the left direction (the length is arbitrary… long enough from the starting point). From the

end point of the arc, start new line and end it at the center point of circles. Now it should be like this pic below.
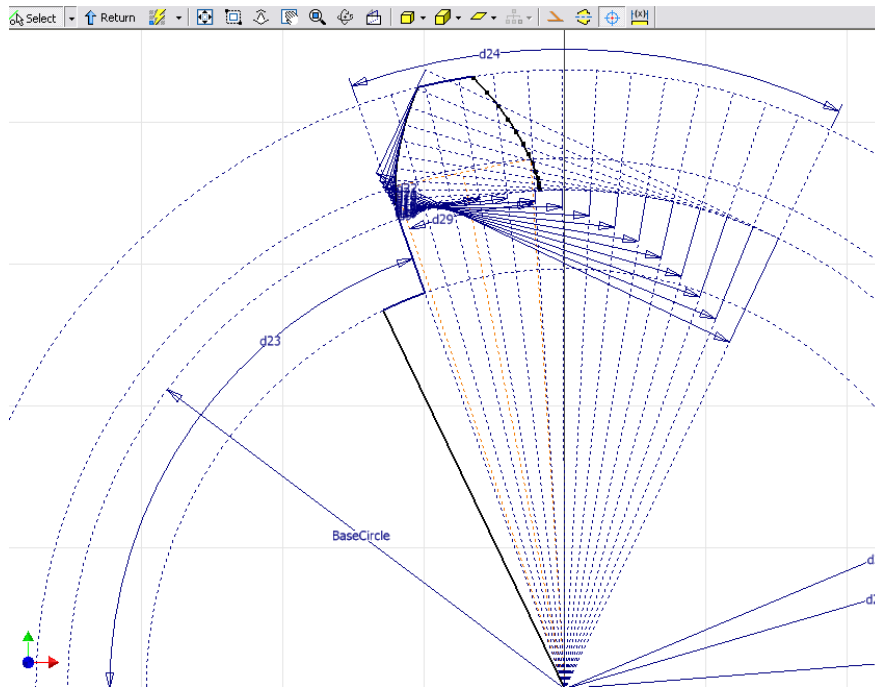

10.) Now mirror the last three lines we've created, with line that we've used to mirror the involute profile. I prefer to add the fillet (at the corner edge between the base cricle and upper line) when doing in part features mode rather than in sketch mode.

11.) This is the exciting part… make it 3D! Yeah… extrude the profile you've made, fill **faceWidth** param in the depth cell. I like 2 ways direction while extruding sketch. Now add 0.3 mm fillet on the corner edge. Now it should be look like this
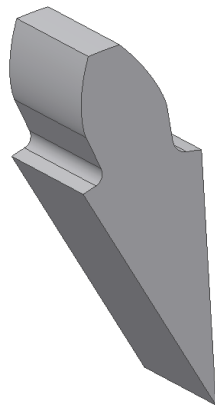
12.) Array Em ! Array the teeth (or tooth?) profil with circular pattern. Fill the Occurrence Count with **nTeeth** param (don't confuse it with filling **nArray**), 360 deg in Occurrence Angle, do not forget to set the Positioning Method to fitted (default setting).
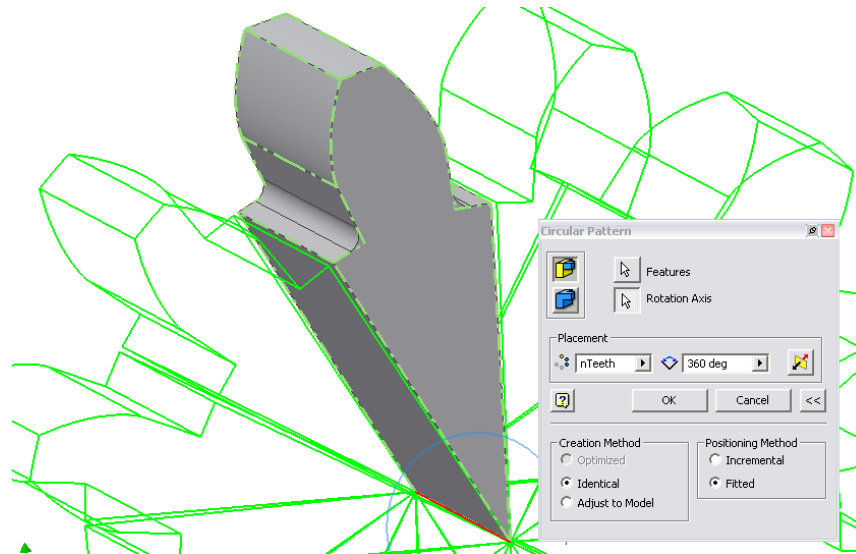
13.) The last resort… make new sketch on the ones of the plane face of the profile. Create circle, set the dimension to **baseCirc_dia** (or coincident constrain it to ones of the corner edge point of base circle… this have the same effect, but I like to do the last one, since it wont add new dim/parameter) then extrude it with **faceWidth** filled on the Depth cell. Try to choose the right diretion.
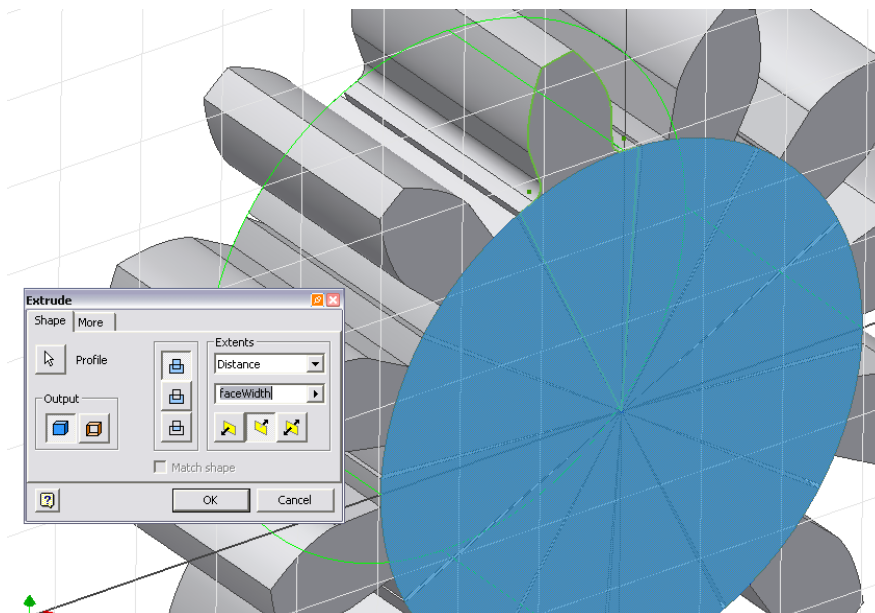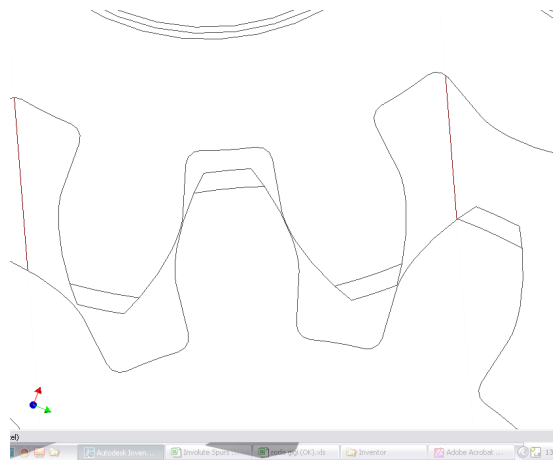
This should be the looks of the gear we've created.
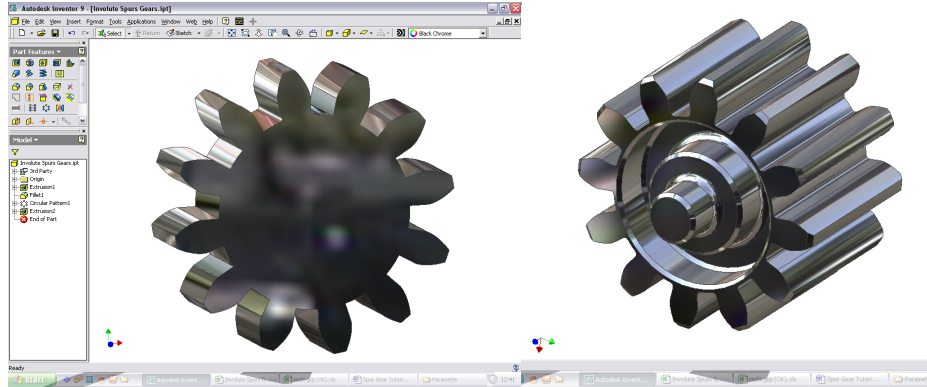
You can add the details so it'll looks like the pic on the right side.

(I wonder if there's any gear which is not using key shaft like that… what…? there is ? fiuhh… what a relief…).

Position pair of the gear we've made properly and constrain em with rotational constrain of 1:1 ratio… and you will see how smooth their face contact.

Note: Because the lack of precission of the calculations I did, the center distance should be added from 15 mm to 15,015 mm to avoid interference.