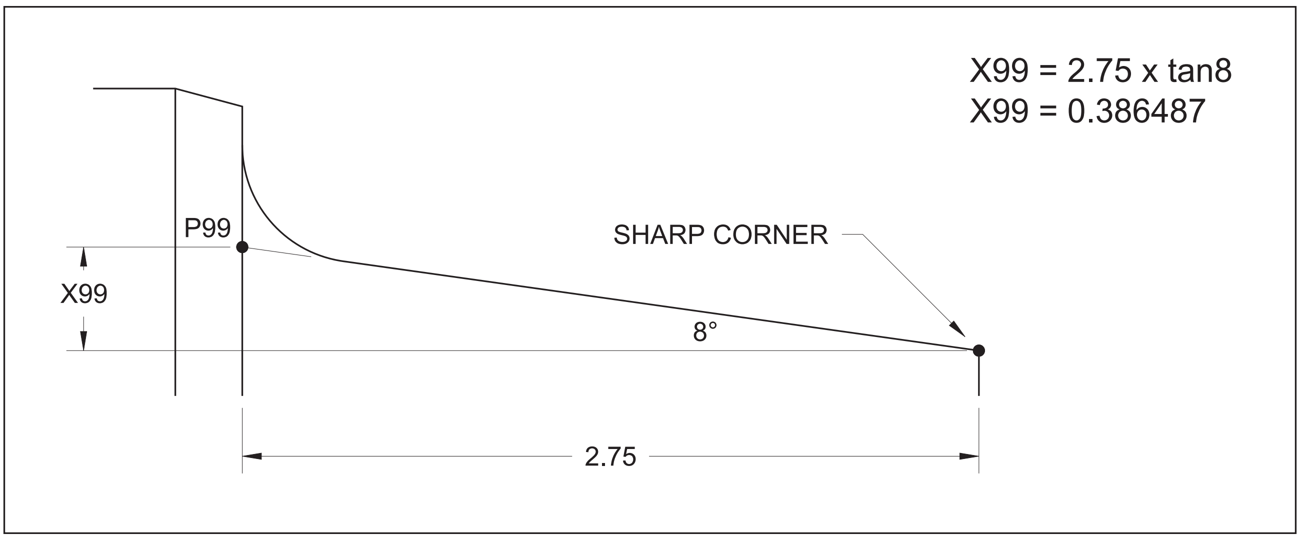
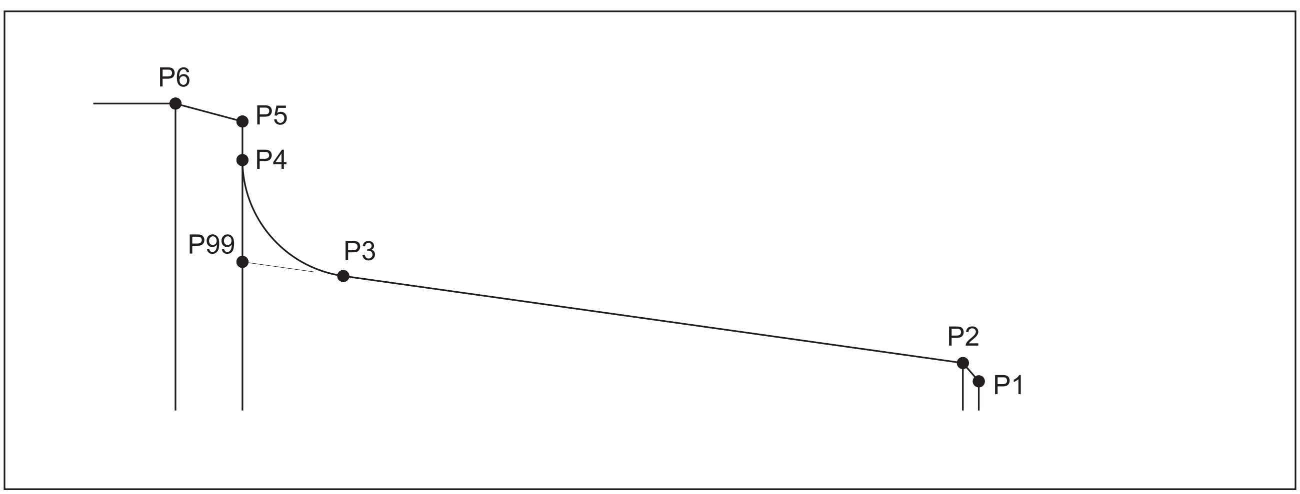
35-02 Solution

Tapered Shaft End

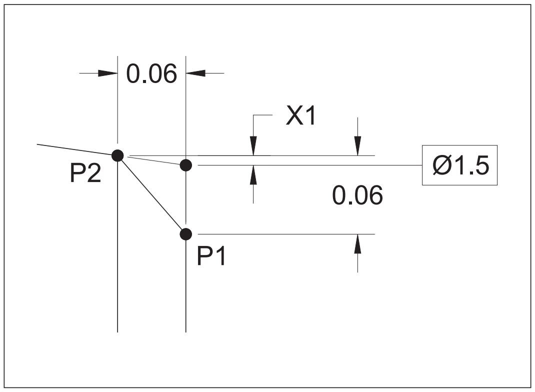
In this project, the face cut will be done first, as in most turning jobs of this type. The G71 cycle will be used to remove bulk of the material with tool T01, and G70 will be used to complete the part to drawing dimensions with tool T03. As only the shaft end is machined, either tool will start off the front chamfer, at an exact point that has to be calculated, and complete the contour just past the 0.25x15° chamfer, at a point which also has to be calculated.

The *first step* towards the solution is to identify the points that have to be calculated. Often, that can be a multi-step process, depending on the complexity of the part geometry. For this example, the contour points related to the taper have to be identified and calculated first, followed by the actual contour change points for the whole contour, as shown in the following illustrations:





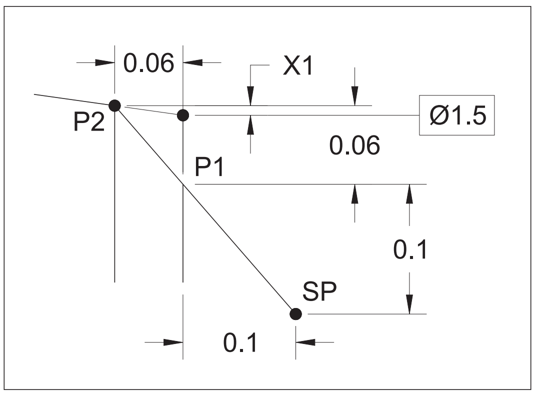
In the *second step*, the contour related calculations can take place. The first calculation will be at the front of part, finding the points P1 and P2 as per drawing:



The ∅1.5 is dimensioned in the drawing to the sharp corner (intersection) of the taper and the front face. This point is only for calculations, and is *not* a contour point. To calculate the XZ coordinates of P1 and P2, the dimension X1 has to be known:

X1 = 0.06 × tan8 = 0.008432

The second chamfer related calculation solves the XZ coordinates of the actual start point *SP*, as they will appear in the part program:

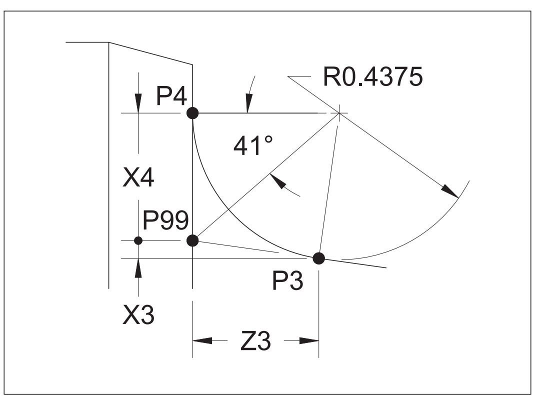


The start point *SP* is 0.1 off the part in both X and Z, as measured from the point P1. To calculate the XZ coordinates, just collect the available data:

SP = ∅1.5 + 2 × X1 - 2 × 0.06 - 2 × 0.1

SP = X1.1969 Z0.1

The points related to the transition between the taper end and the 0.4375 blend radius have to be calculated next:



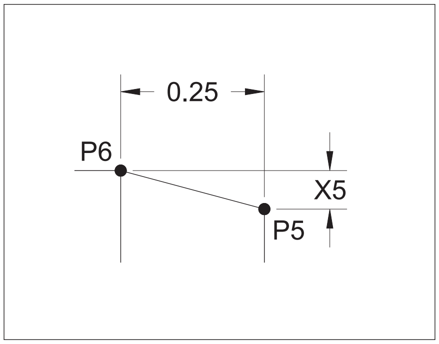
The X4 dimension must be calculated first, because it will be used to calculate dimensions X3 and Z3:

X4 = 0.4375 × tan41 = 0.380313

X3 = X4 × sin8 = 0.052929

Z3 = X4 × cos8 = 0.376612

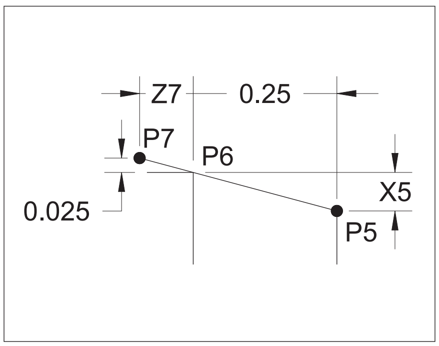
At the end of the contour, the tool exits on a chamfer. Two final calculations will be necessary - one is to find the dimension X5 of the chamfer itself, the other is to find the dimension Z7 which extends the chamfer:



This calculation will not be used in the program itself, but is necessary to calculate point P7, which is 0.025 above the final diameter:

X5 = 0.25 × tan15 = 0.066987

Point P7 is the actual end point of the contour itself, followed by a lead-out motion, away from the diameter. To calculate P7 point, it is necessary to determine how far above the diameter the tool will end.



In the example, the amount is 0.025 per side. Now, the final dimension Z7 can be calculated:

Z7 = 0.025 / tan15 = 0.093301

The part appears fairly simple in the drawing, but the actual number of calculations brings some challenge to the project. The final step is, of course, to develop the CNC program, based on the calculations just developed.

Study all previous steps first - the XZ coordinates in the CNC program are the direct result of using all calculated dimensions, applied to the CNC program.

The first version of the program uses G71 cycle as a single block input:

```
(35-02 TAPERED SHAFT END - ONE BLOCK FOR G71)
(EXTEND PART MINIMUM OF 3.25 FROM THE FACE OF JAWS)

(T01 - 80 DEG OD TOOL)
N1 G20                                  (ENGLISH UNITS SELECTED)
N2 T0100 M41                            (TOOL 1 AND LOW GEAR RANGE)
N3 G96 S450 M03                         (CONSTANT SURFACE SPEED IS 450 FPM)
N4 G00 G41 X3.7 Z0 T0101 M08            (START POINT FOR FACING)
N5 G01 X-0.07 F0.008                    (FACING CUT)
N6 G00 Z0.1                             (CLEAR OFF PART)
N7 G42 X3.55                            (CONTOUR START POINT AT X3.55 Z0.1)
N8 G71 P9 Q15 U0.06 W0.004 D1750 F0.016 (ROUGHING CYCLE)
N9 G00 X1.1969                          (SP POINT)
N10 G01 X1.5168 Z-0.06 F0.002           (P2 POINT)
N11 X2.1672 Z-2.3734 F0.006             (P3 POINT)
N12 G02 X3.0336 Z-2.75 R0.4375          (P4 POINT - I0.4332 K0.0609 INSTEAD OF R)
N13 G01 X3.322                          (P5 POINT)
N14 X3.506 Z-3.0933 F0.004              (P7 POINT - 0.025 ABOVE STOCK DIA)
N15 U0.2 F0.03                          (LEAD-OUT CLEARANCE)
N16 G00 G40 X10.0 Z4.0 T0100 M05        (TOOL INDEXING POSITION)
N17 M01                                 (OPTIONAL STOP)
```

```
(T03 - 55 DEG OD TOOL)
N18 T0300 M42                          (TOOL 3 AND HIGH GEAR RANGE)
N19 G96 S600 M03                       (CONSTANT SURFACE SPEED IS 600 FPM)
N20 G00 G42 X3.55 Z0.1 T0303 M08       (START POINT FOR FINISHING)
N21 G70 P9 Q15                         (FINISHING CYCLE)
N22 G00 G40 X10.0 Z4.0 T0300 M09       (TOOL INDEXING POSITION
N23 M30                                (END OF PROGRAM)
%
```

The second version of the program uses G71 cycle as a double block input - the block numbers and the P and Q block identifiers have shifted (roughing and finishing), because of the two-block method of G71 entry:

```
(35-02 TAPERED SHAFT END - TWO BLOCKS FOR G71)
(EXTEND PART MINIMUM OF 3.25 FROM THE FACE OF JAWS)

(T01 - 80 DEG OD TOOL)
N1 G20                                 (ENGLISH UNITS SELECTED)
N2 T0100 M41                           (TOOL 1 AND LOW GEAR RANGE)
N3 G96 S450 M03                        (CONSTANT SURFACE SPEED IS 450 FPM)
N4 G00 G41 X3.7 Z0 T0101 M08           (START POINT FOR FACING)
N5 G01 X-0.07 F0.008                   (FACING CUT)
N6 G00 Z0.1                            (CLEAR OFF PART)
N7 G42 X3.55                           (CONTOUR START POINT AT X3.55 Z0.1)
N8 G71 U0.175 R0.05                    (ROUGHING CYCLE - BLOCK 1 OF 2)
N9 G71 P10 Q16 U0.06 W0.004 F0.016     (ROUGHING CYCLE - BLOCK 2 OF 2)
N10 G00 X1.1969                        (SP POINT)
N11 G01 X1.5168 Z-0.06 F0.002          (P2 POINT)
N12 X2.1672 Z-2.3734 F0.006            (P3 POINT)
N13 G02 X3.0336 Z-2.75 R0.4375         (P4 POINT - I0.4332 K0.0609 INSTEAD OF R)
N14 G01 X3.322                         (P5 POINT)
N15 X3.506 Z-3.0933 F0.004             (P7 POINT - 0.025 ABOVE STOCK DIA)
N16 U0.2 F0.03                         (LEAD-OUT CLERANCE)
N17 G00 G40 X10.0 Z4.0 T0100 M05       (TOOL INDEXING POSITION)
N18 M01                                (OPTIONAL STOP)

(T03 - 55 DEG OD TOOL)
N19 T0300 M42                          (TOOL 3 AND HIGH GEAR RANGE)
N20 G96 S600 M03                       (CONSTANT SURFACE SPEED IS 600 FPM)
N21 G00 G42 X3.55 Z0.1 T0303 M08       (START POINT FOR FINISHING)
N22 G70 P10 Q16                        (FINISHING CYCLE)
N23 G00 G40 X10.0 Z4.0 T0300 M09       (TOOL INDEXING POSITION
N24 M30                                (END OF PROGRAM)
%
```

One of the conditions specified for the project was to indicate the minimum setup extension of the part from the face of jaws. Theoretically, the minimum extension is the last Z-coordinate in the program, which is Z-3.0933. Any reasonable amount *further* than that location can be considered as correct. In the program, the 3.25 extension amount was selected as reasonable and practical.

This concludes a fairly involved CNC lathe programming project. The process shown in this project can be very easily adapted to many similar applications.