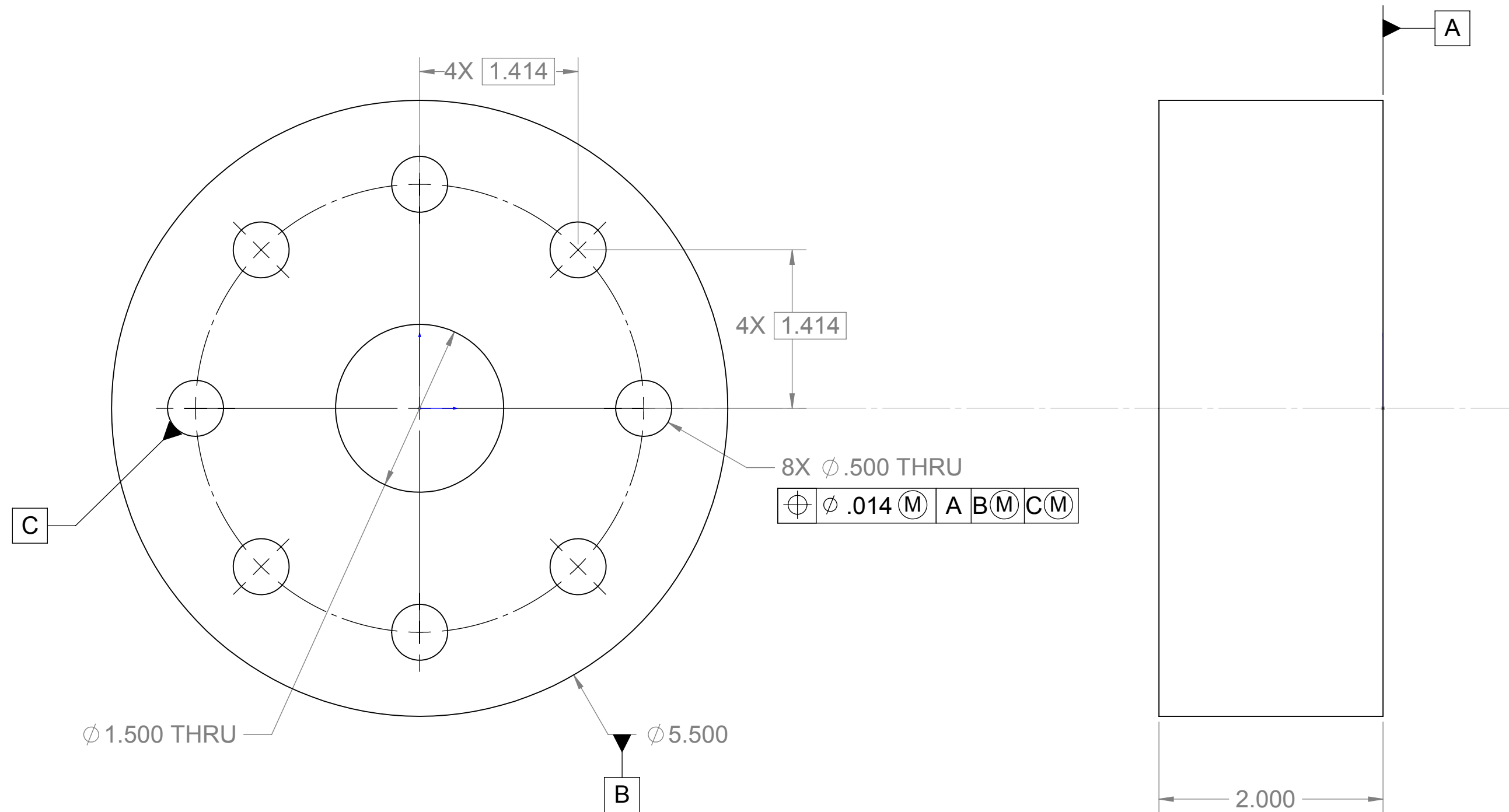


FACT FROM THE DRAFTING REQUIREMENTS MANUAL 11TH EDITION:
ENGINEERING DRAWINGS ARE REQUIREMENTS. THEY DO NOT TELL
YOU HOW TO MACHINE THE PARTS. GD&T IS AN INSPECTION TECHNIQUE USED TO
CHECK THE REQUIREMENT OF THE END PRODUCT. THERE ARE NO DATUM PLANES
ON THE DRAWINGS TO BE CHECKED. JUST DATUM FEATURES THAT REPRESENT THE
DATUM REFERENCE FRAME OF THE 1ST, 2ND & 3RD PLANES.

ASME Y14.5 2009

"THERE IS MORE THAN 1 WAY TO SKIN A CAT WHEN DETAILING DRAWINGS".
BUT THERE IS NO OTHER VARIATIONS TO A STANDARD OF ASME Y14.5 2009 .

YOU CAN DIMENSION THIS DRAWING HOW MANY WAS YOU WANT BUT THE DATUM FEATURES WILL NOT CHANGE.



QUESTION: WILL SOME AGREE THAT THE DATUM FEATURE C IN THE POSITIONAL TOLERANCE FRAME APPLIES AS A 3RD REFERENCE. FOR THIS HOLE PATTERN.

REASON: IT IS USED FOR CLOCKING. SO IT LOCKS DOWN DATUM FEATURE B IN THE 2ND AND 3RD PLANES OF THE DATUM REFERENCE FRAME. TRULY RESTRICTING THE SIX DEGREES OF FREEDOM.

I HEARD: THROUGH THE GRAPE VINE SOMEONE THINKS THAT YOU CANNOT REFERENCE A DATUM WITHIN ITSELF. WHAT THEY MEANT WAS IT CAN'T BE APART OF THE HOLE PATTERN AND REFERENCING ITSELF IN THE POSITIONAL TOLERANCE FRAMES $8 \times$. NOTICE THEY USED THE TERM DATUM.

MY ANSWER: HERE I AM SIMPLY SAYING TO THE INSPECTOR LOCATE DATUM FEATURE C.

OLD SCHOOL: YOU DIMENSION OUT TO THE DATUM PLANE.

THE FOR GOTTEN OLD SCHOOL WHICH WAS NEVER NEW SCHOOL: YOU SELECT DATUM FEATURES.