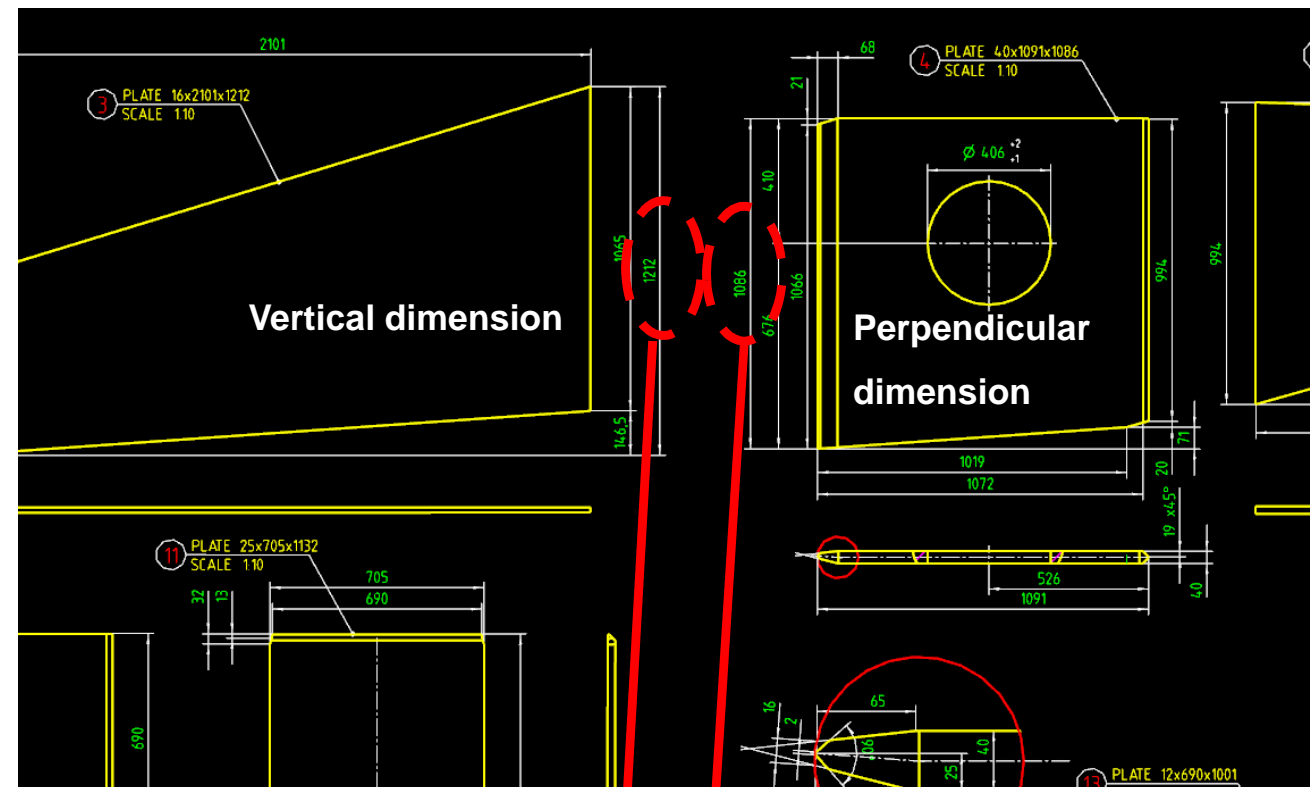## NX draft

Drawing defaults in NX drafts:
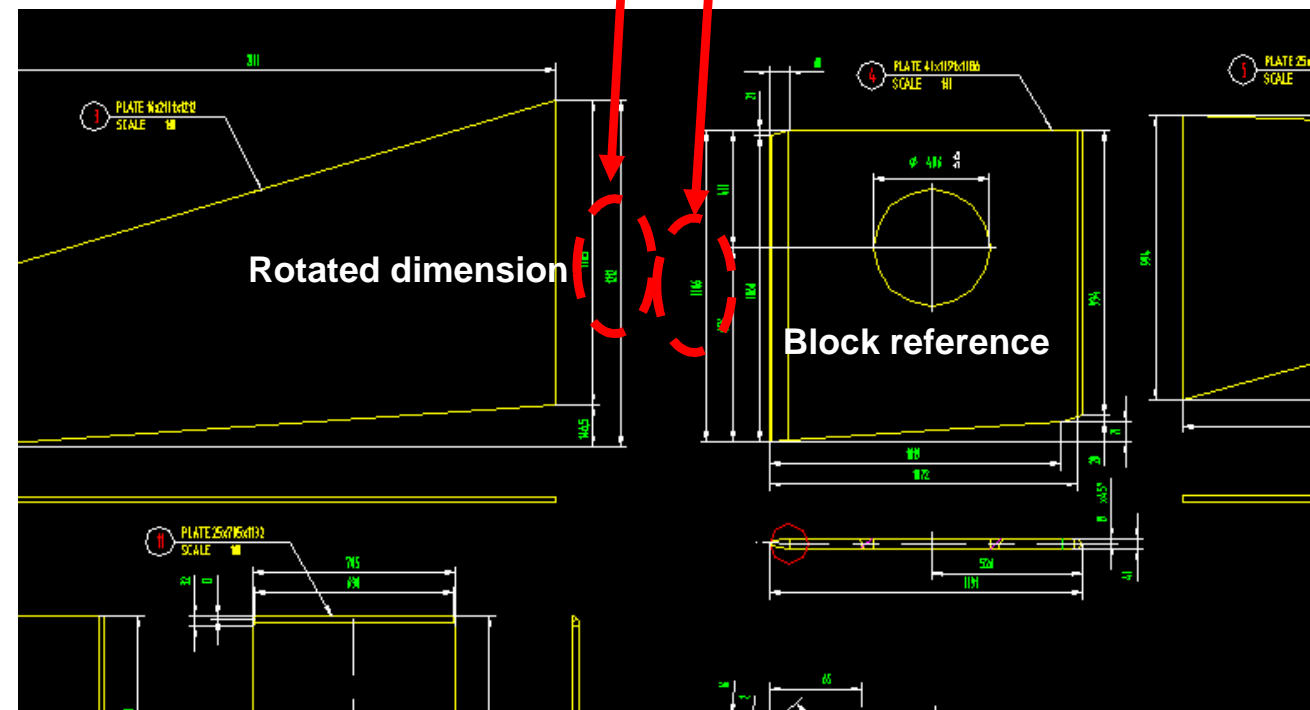
Vertical dimensions always are corrctly converted into rotated dimensions in Acad.

Perpendicular dimensions always convert into blocks in Acad.

For the upcoming drawing and modelling standard (common wow in NX) **avoid** the use of perpendicular dimensions!

Vertical dimension

Perpendicular dimension

Acad dwg

Rotated dimension

Block reference

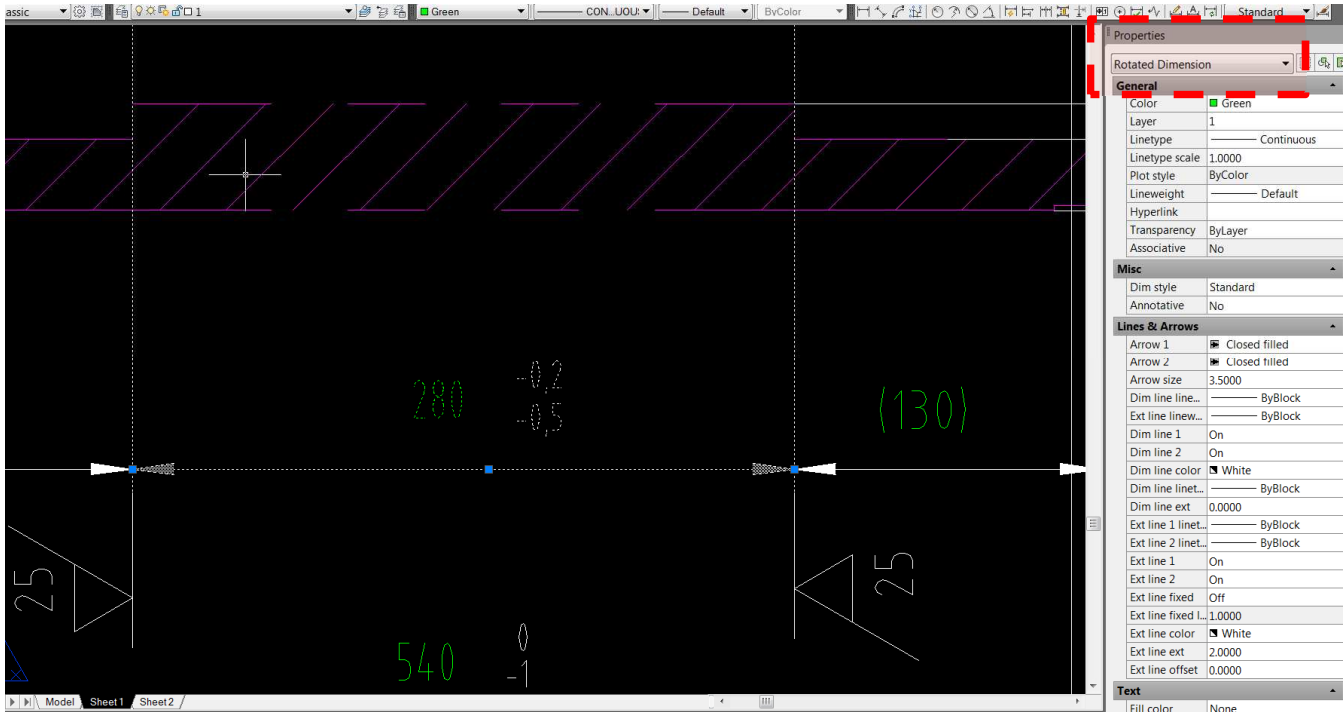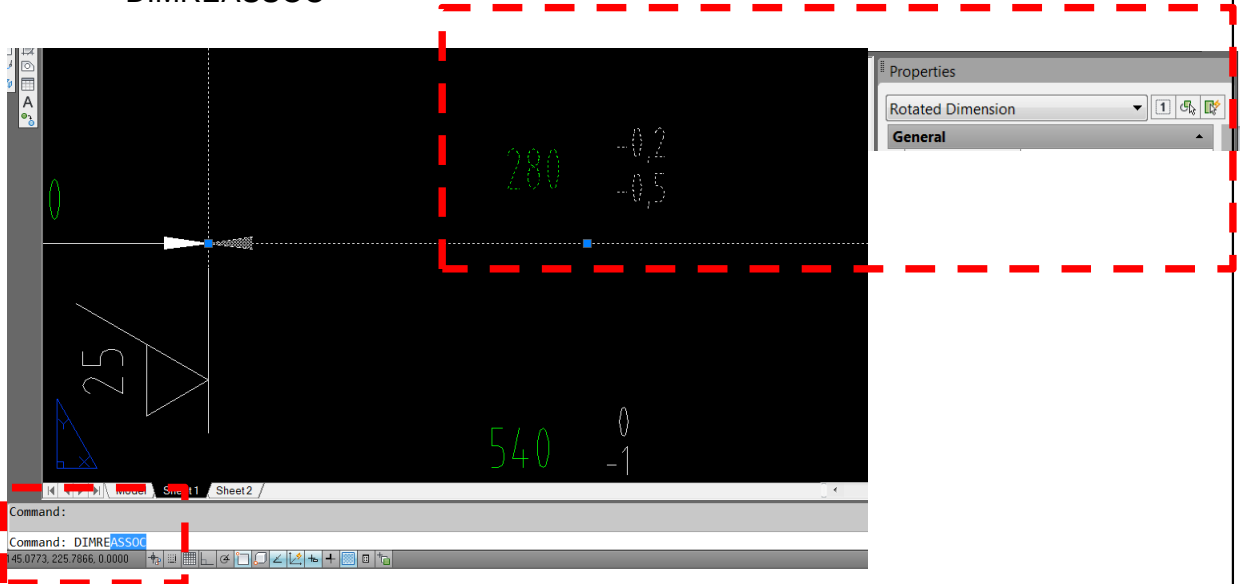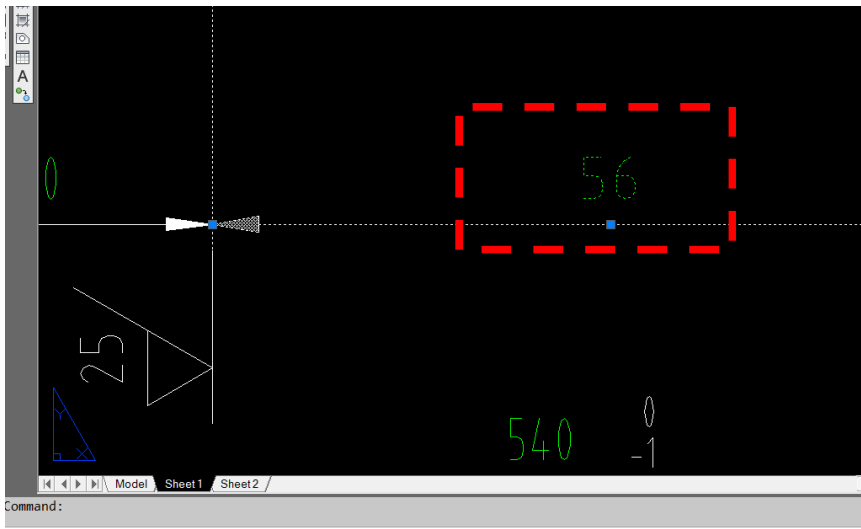|	Teamcenter / NX **DWG-Export from NX**			
SANDVIK	**Standardisation - Modularisation**			
Sandvik Mining and Construction Materials Handling GmbH & Co KG	Author:J.Schütz	Date: 19.04.2013 Status: FINAL	Rev.	0

1.) after converting, the dimension is still a dimension,
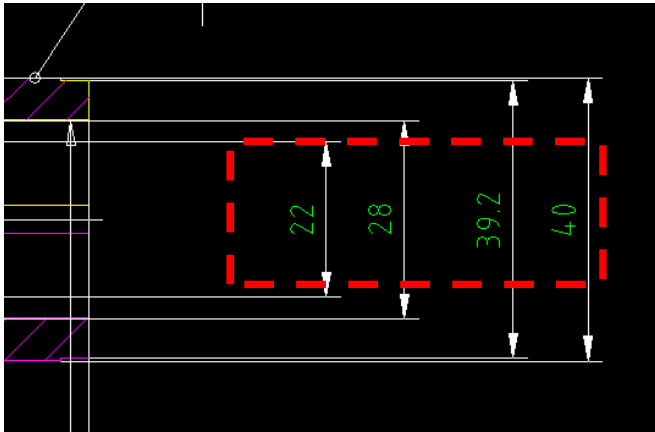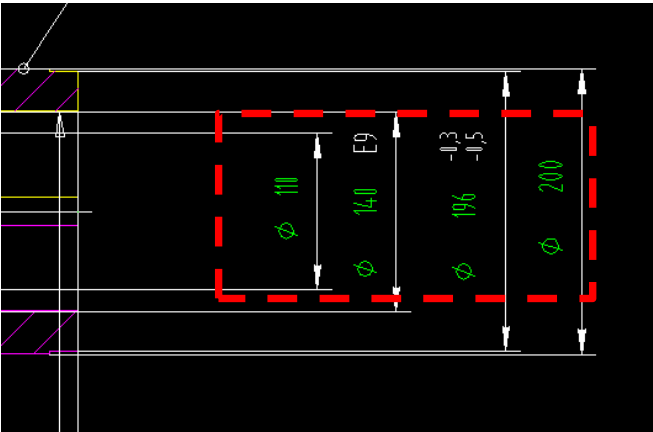   but not associative

2.) activate all rotated dimensions and perform
   DIMREASSOC

3.) dimension is associative again, but tolerances are
   lost and scale has changed to 1:1

4.) Diameter informations and tolerances also get lost
   after making them reassociative

Teamcenter / NX
DWG-Export from NX, default settings

Standardisation - Modularisation

| Sandvik Mining and Construction Materials Handling GmbH & Co KG | Author:J.Schütz | Date: 12.04.2013 Status: FINAL | Rev. | 0 |