Technical Drawing

3D

step by step
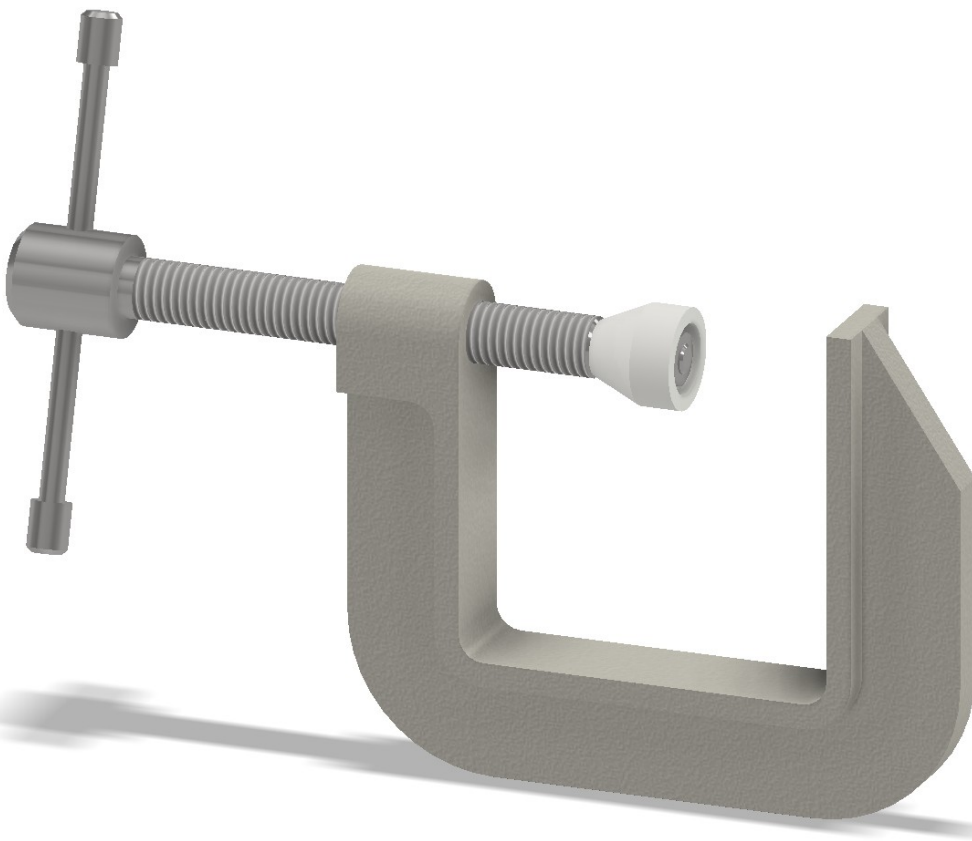
AUTODESK
**Inventor Professional**

SUMMARY

WORK FOLDERS

Before starting to create a 3D model of a structure/machine, it is necessary to create a work folder on the PC's hard disk where the drawing will be saved. We always recommend working on the PC's hard disk and not on USB sticks, which are 10 to 100 times slower in reading and writing operations!!!

The quickest way is to press the 'windows flag' key and the 'E' key together so that the windows *file explorer* opens on the screen.

 + E
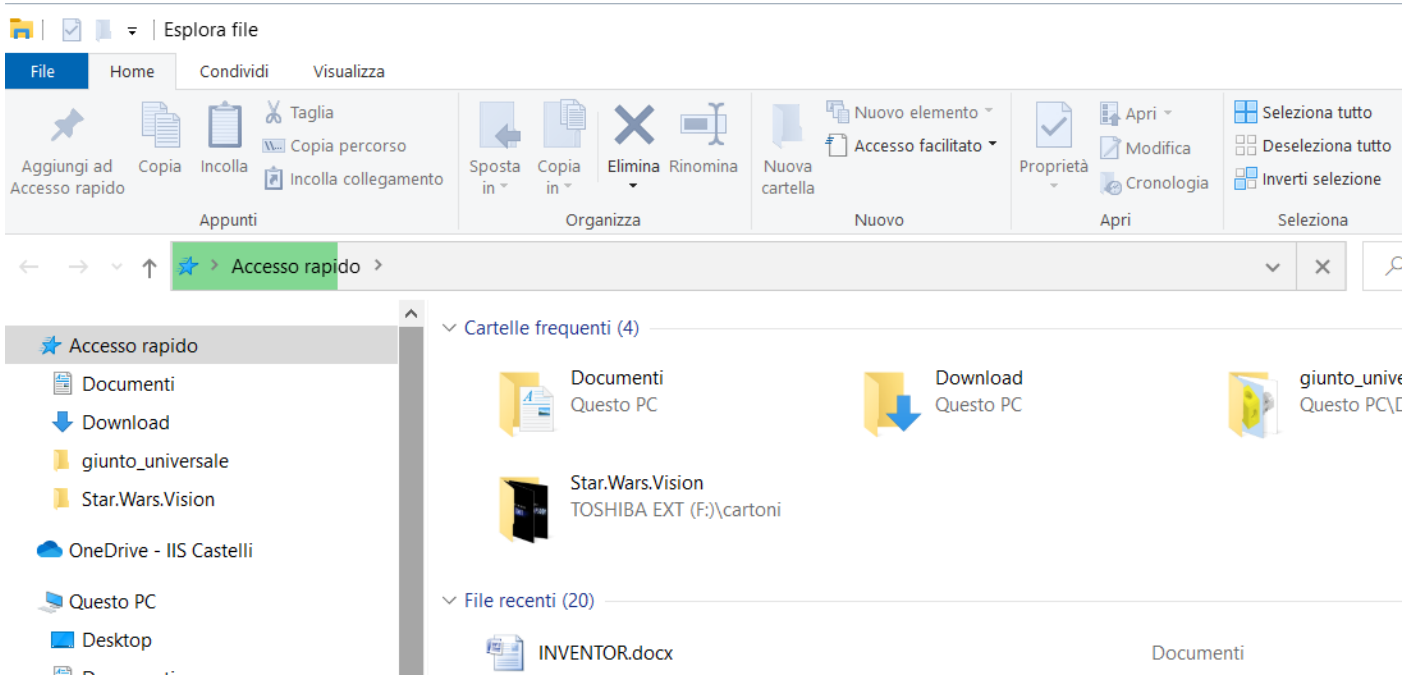
You must now enter the 'documents' folder and then create the work folder by clicking on 'new folder'.
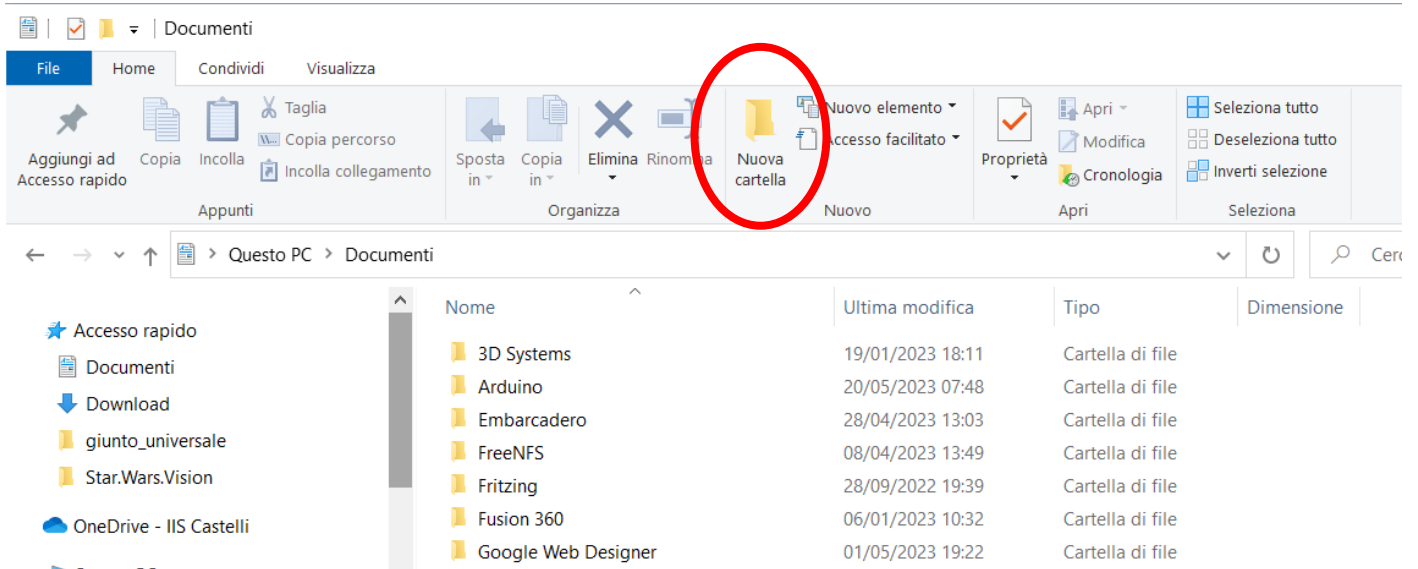

Assign the workbook a name that clearly identifies the design to be created.

In order to share a 3D project, it is necessary to create a compressed archive containing all the drawings of the parts that make up the designed structure/machine.

Windows version 8 onwards allows you to create compressed archives without installing specific programmes.

Simply select the project folder and click on zip in the 'share' section of the window.
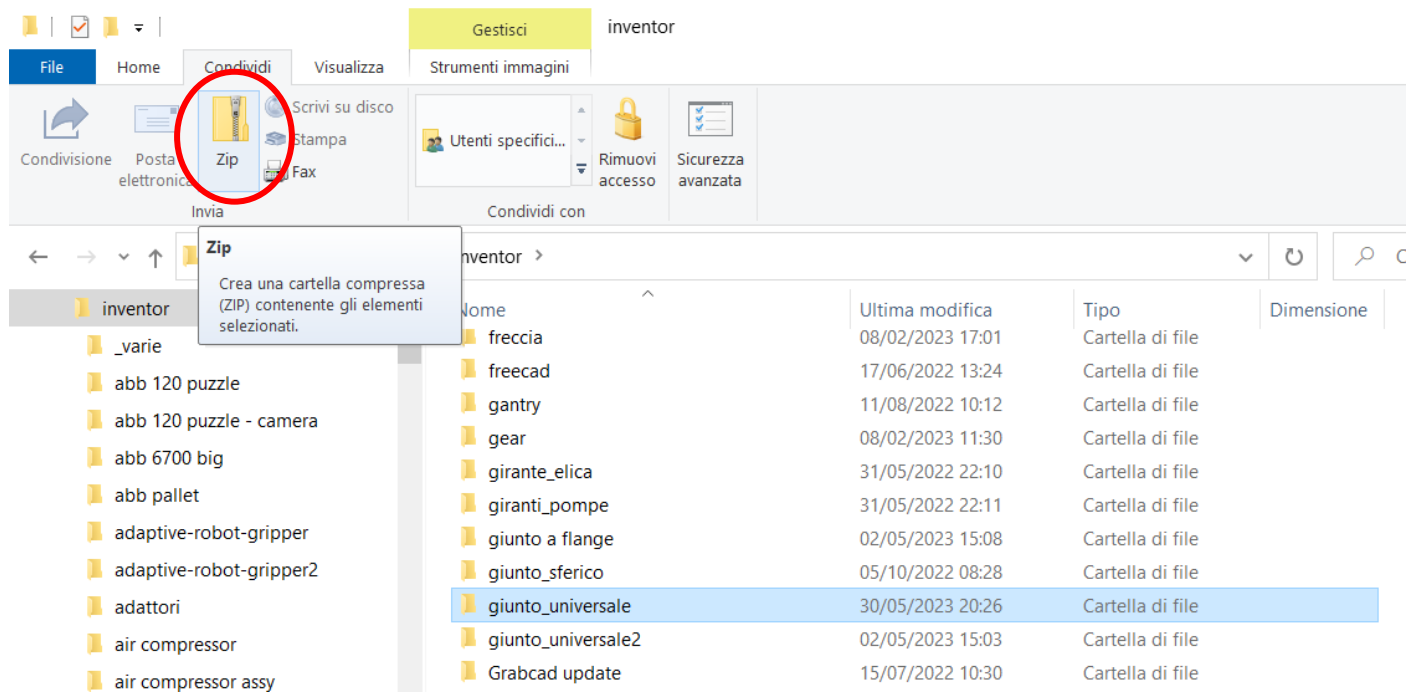
A single file will be created (extension .zip) containing all files in the selected folder.
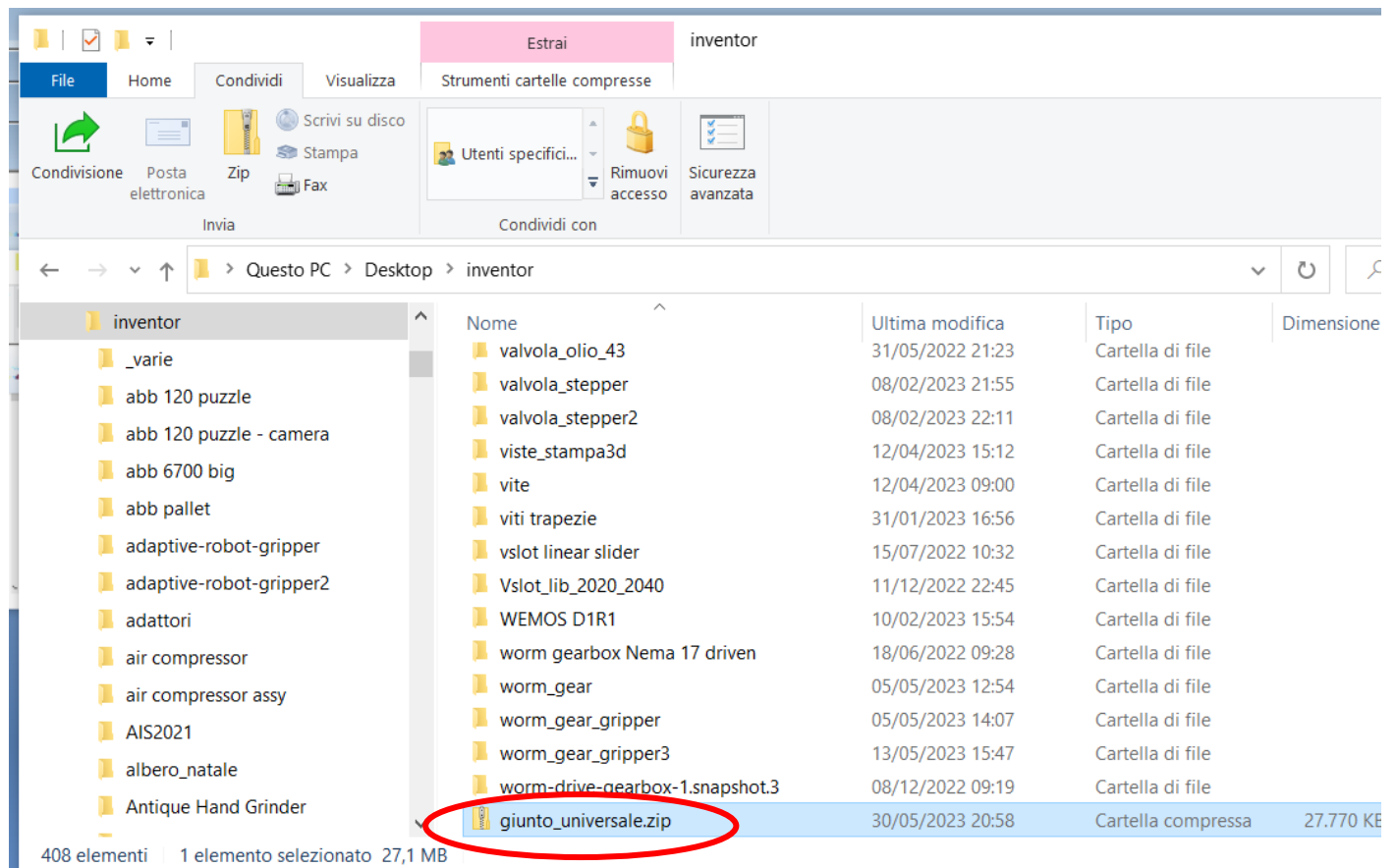
The 3D drawing of a structure/machine in Inventor always starts with the creation of its individual parts.
An assembly is the set of individual parts that are properly connected (constrained) to each other.

For example, the figure below shows the assembly of a vice, which consists of 7 simple parts (note that the nut is available in Inventor's *content centre* and has therefore not been drawn).
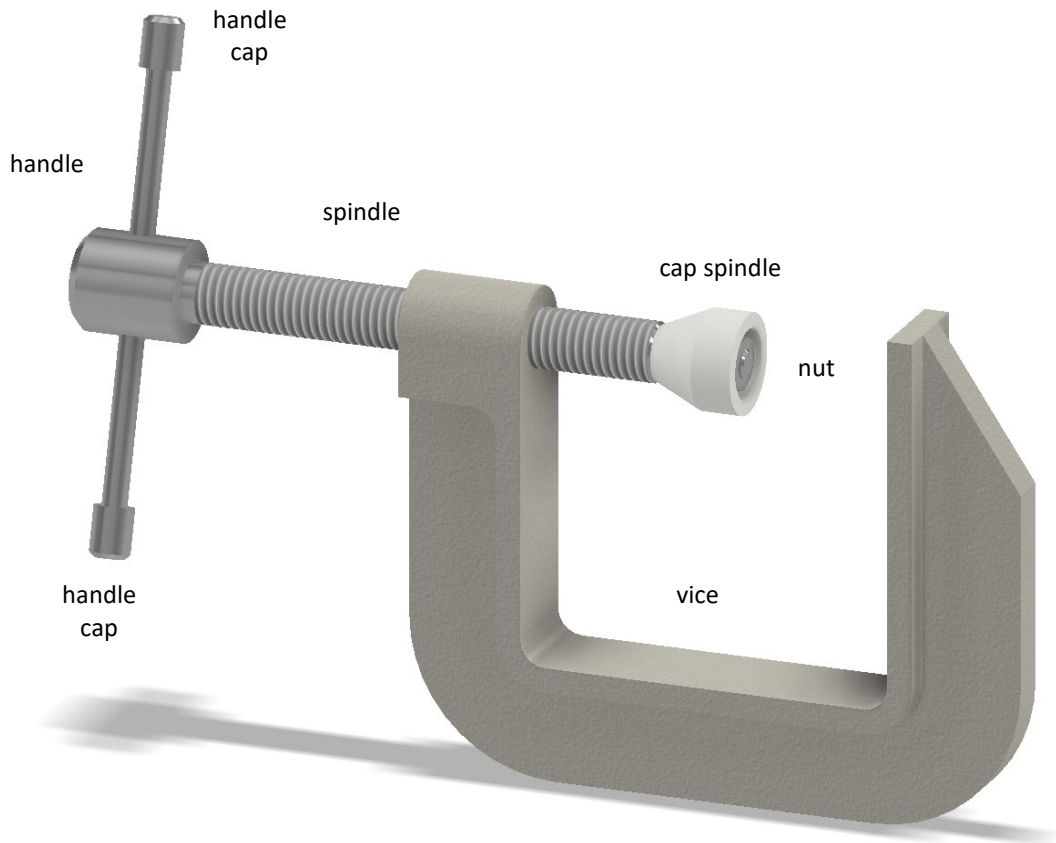
The Inventor tree below shows the various parts of the "vise" assembly available at the link:
http://energiazero.org/disegno/morsa_semplice.zip
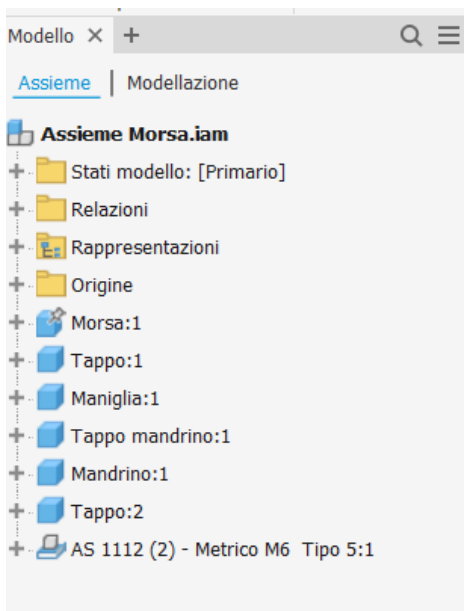
Inventor shows all parts of the assembly in a hierarchical tree structure.

The parts with the *cube icon* are those drawn by the user while those with *a sheet icon drawn from the* drawer are those available in Inventor's drawing libraries.

Inventor provides a large library of basic, commercially available mechanical components (bolts, bearings, etc.) that save the user from having to design every single standardised part.

Before arriving at the construction of an assembly, it is therefore necessary to draw all the individual parts that constitute it.

To create a part, click on the ' *new file* menu' to open the individual PART creation mask.
Select "standard (mm).ipt".   (the extension .ipt identifies a part)
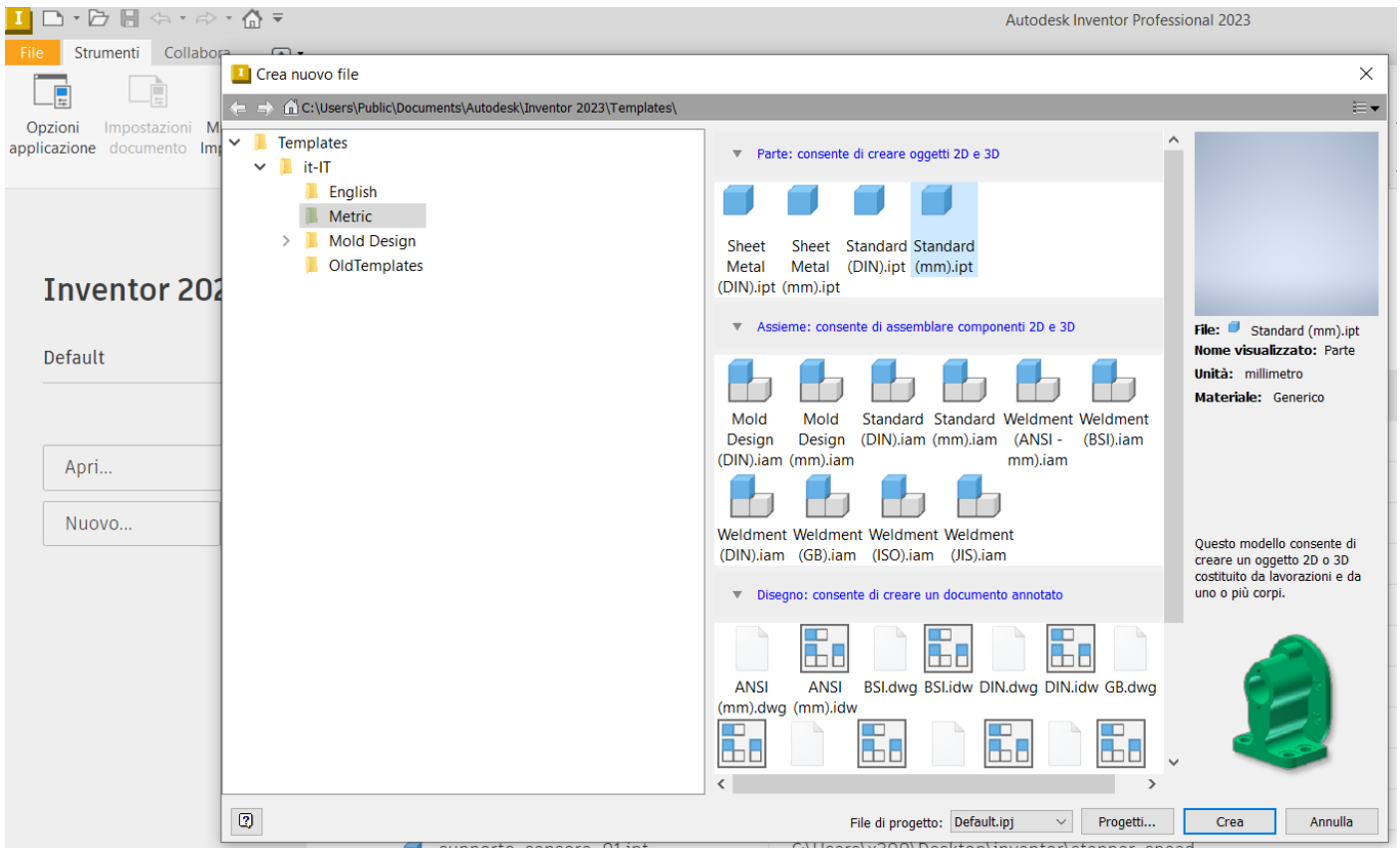
Immediately after confirming with 'Create' you must save the file (format .ipt) with the name of the part you are creating (take care to choose a unique name) in the project folder that will contain all individual parts and the final assembly. Assign the project folder a meaningful name that identifies the structure/machine to be created.

For each individual machine part, except for those available in Inventor's libraries, the above procedure must be repeated. Each unique part will have a corresponding file with the .ipt format in the project folder.
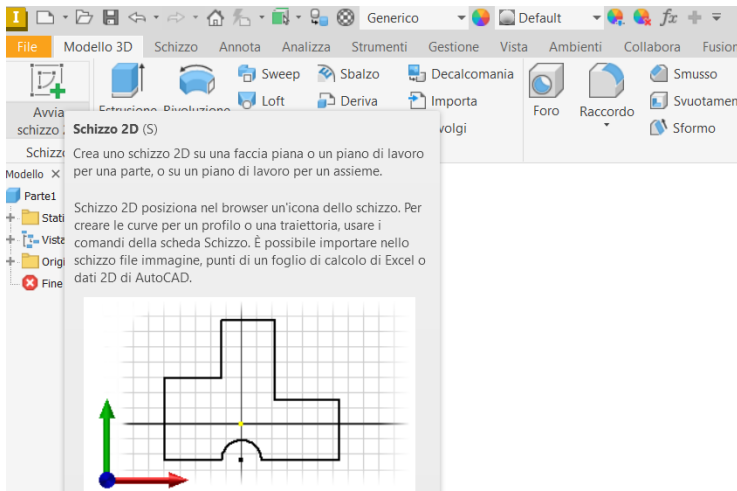When all parts have been drawn, the assembly can be constructed.
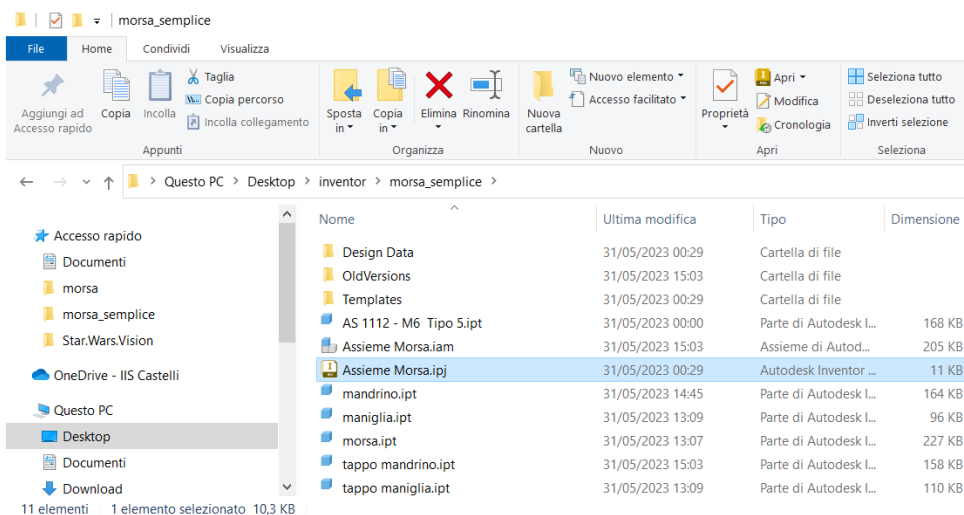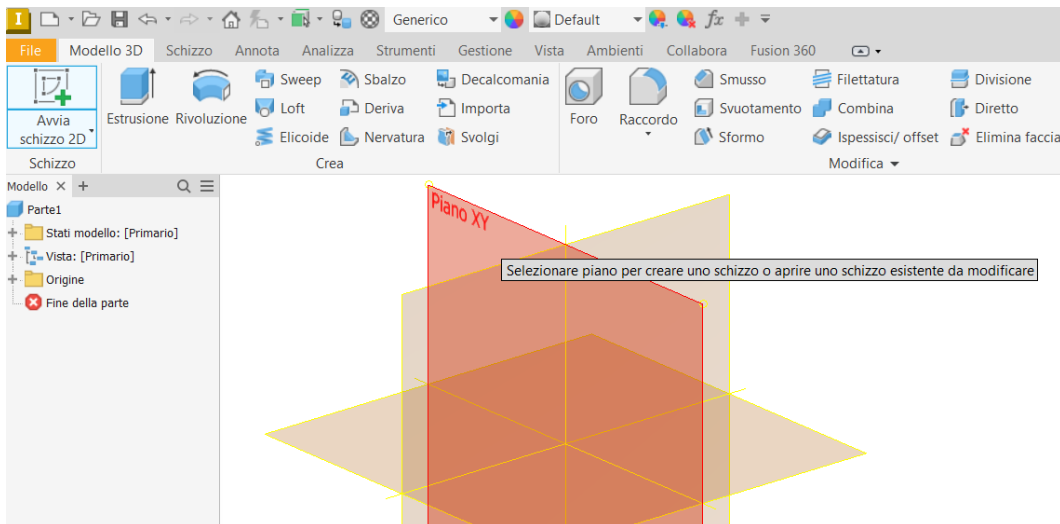

project folder
vice

CREATION OF 2D SKETCHES

The creation of a part always starts with the creation of a sketch, i.e. a 2D drawing, which is then used with the 3D commands (extrusion, revolution, etc.) to obtain a solid.


Clicking on start 2D sketch prompts you to select the work plane on which to start the 2D drawing.

Example of a 2D sketch. Get used to adding dimensions to sketches from the outset by checking whether changing the dimension makes the sketch resize as expected. Inventor is a *parametric* 3D CAD. By changing a dimension, the whole part should change correctly if the dimensions have been assigned correctly.
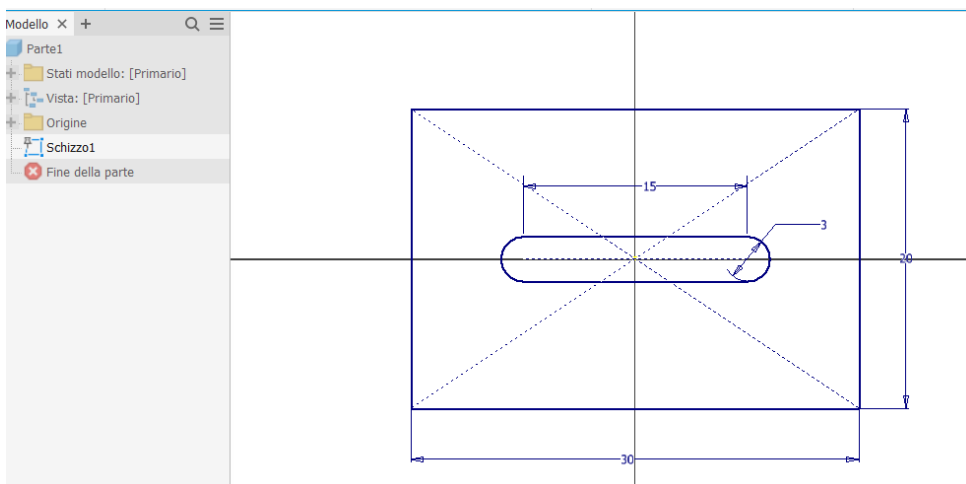
Now you can use the 3D *extrusion* command to obtain a solid from the sketch you have just drawn.
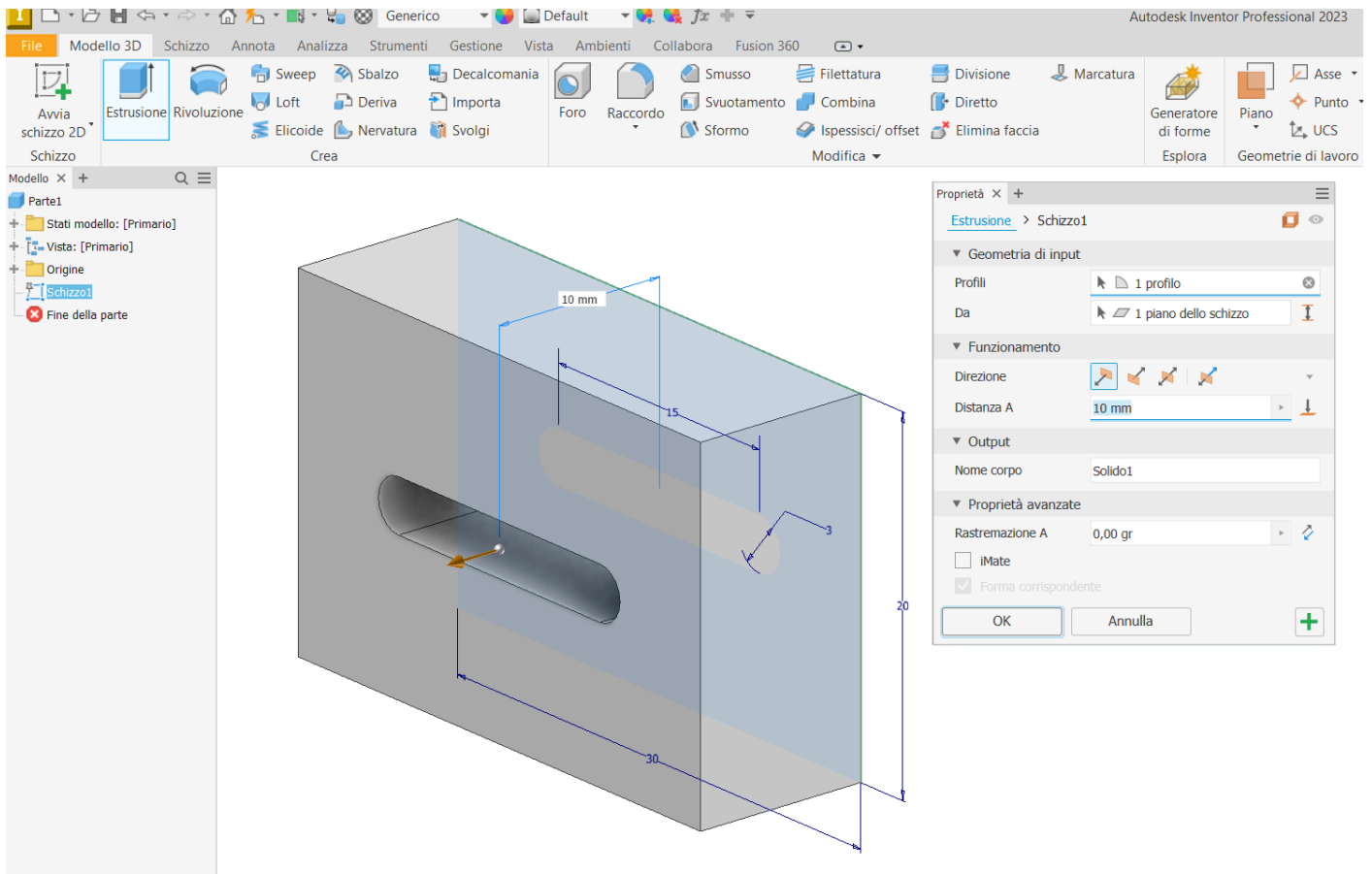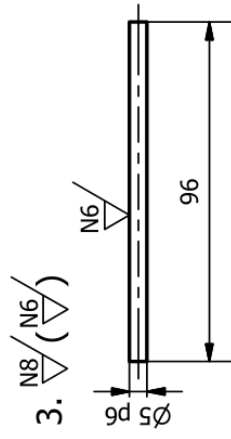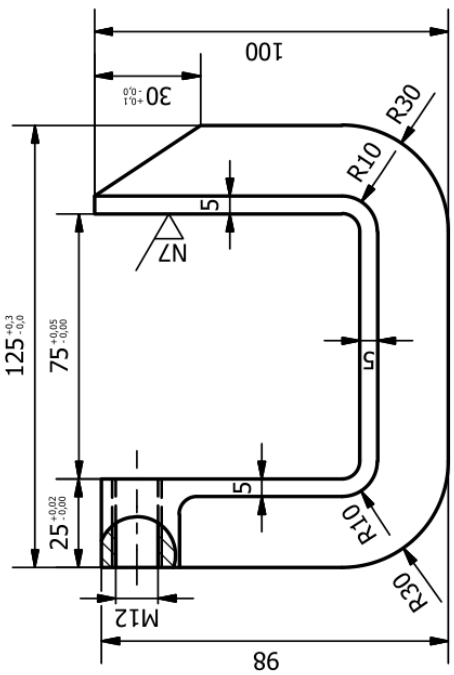

Other 3D commands such as chamfer or fillet can be applied to the resulting solid, or other 2D sketches can be drawn on its flat faces to obtain other machining operations.
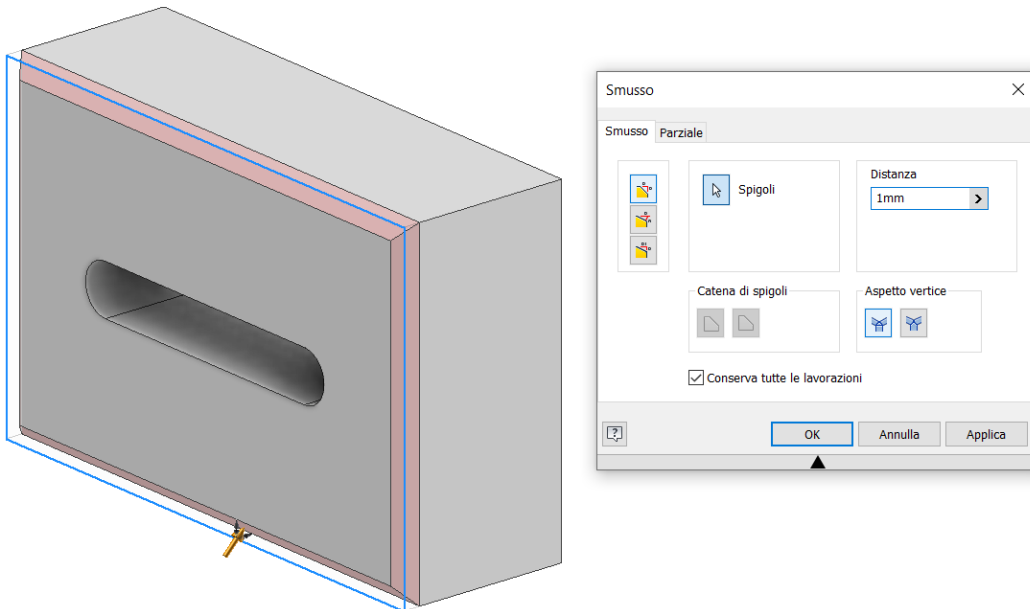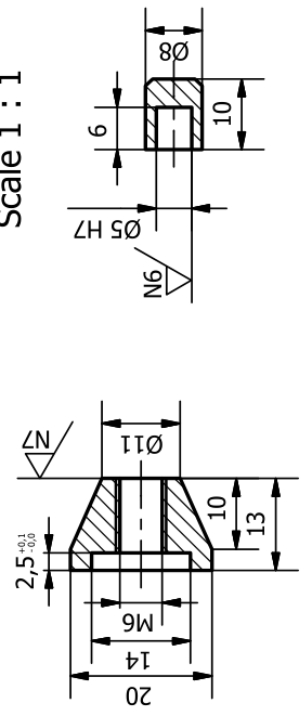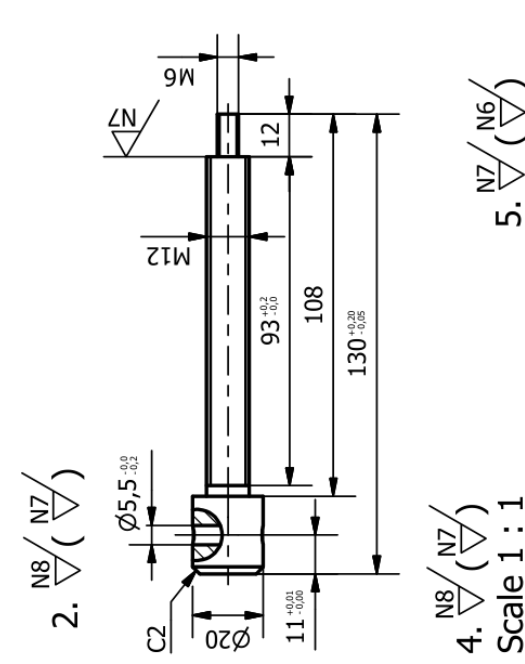
Note:
1. General Tolerance ±0.1
2. General Chamfer 1x45°
3. General Radius R5

PARTS LIST

ITEM	QTY	PART NUMBER	DESCRIPTION
1	1	Handle	Mild Stell / DIA 8x104
2	2	Cup Handle	Mild Stell / DIA 8x10
3	1	Spindle	Mild Stell / DIA 20x130
4	1	Clamp Jaw	Cast Iron / DIA 125x100x20
5	1	Spindle Cap	Mild Stell / DIA 20x13

Skala : 1 : 2
Ukuran : mm
Tanggal : 02-11-2020

A4

Create the following 2D sketch, taking care to use the axes of the sheet as the initial reference.

ORIGIN AXES
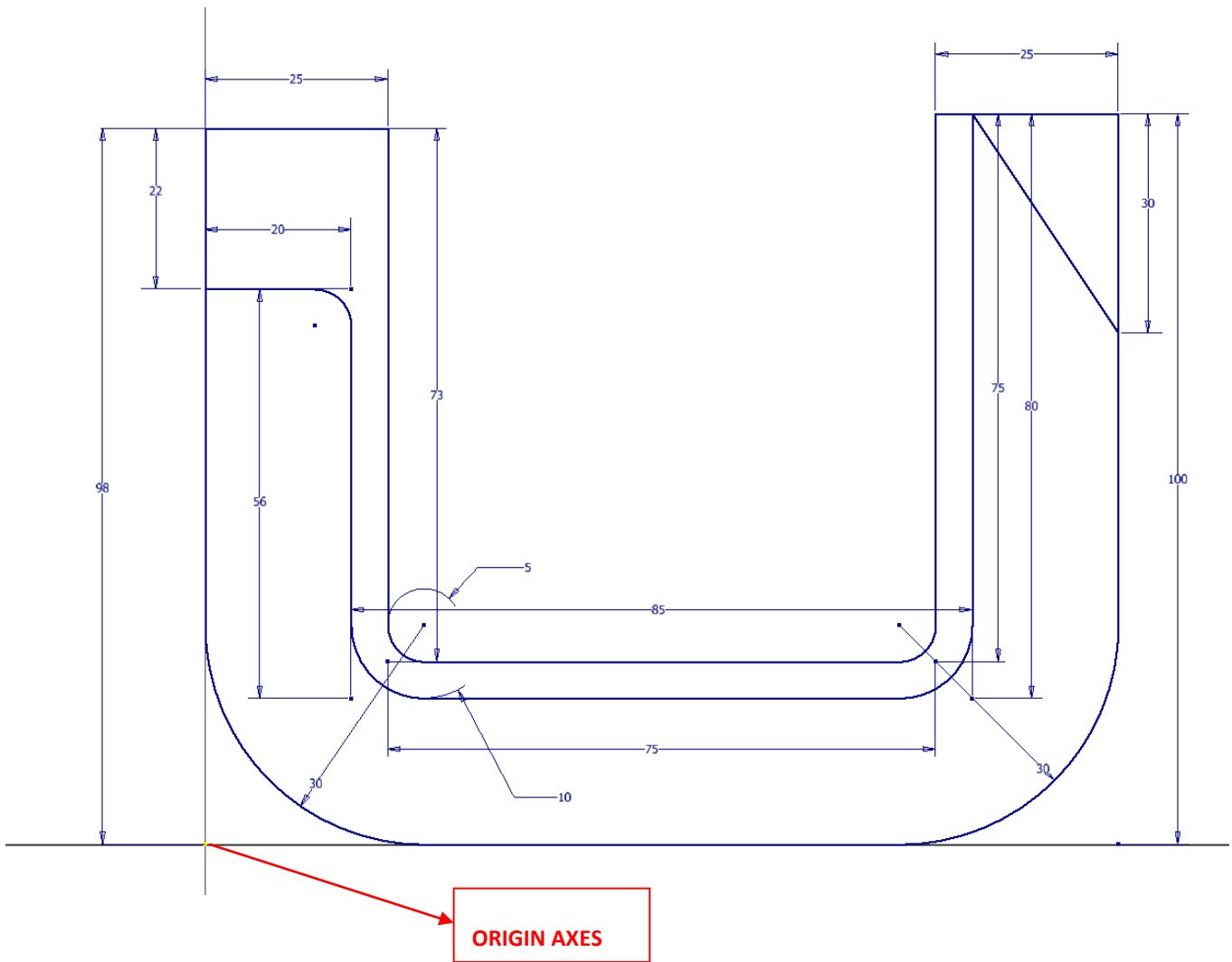
Extrude as shown.
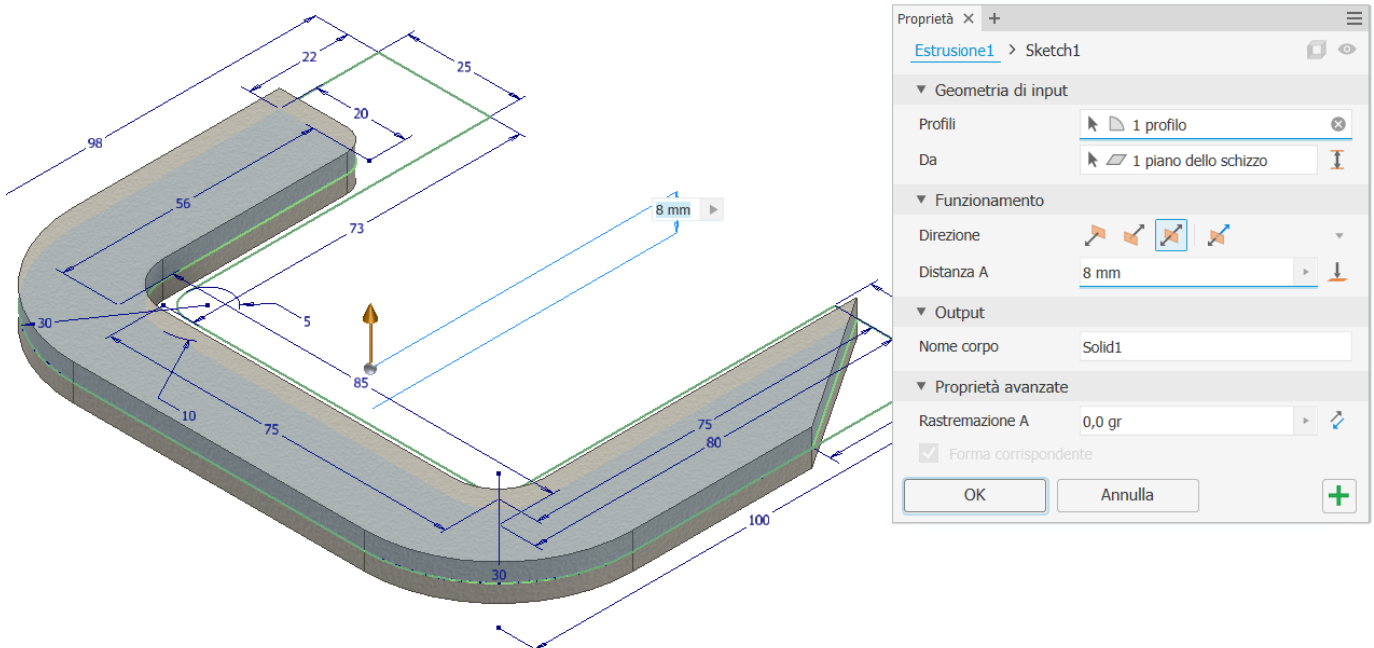
Make the initial sketch visible and perform a second extrusion as in the figure (we use sketch1 twice separately).
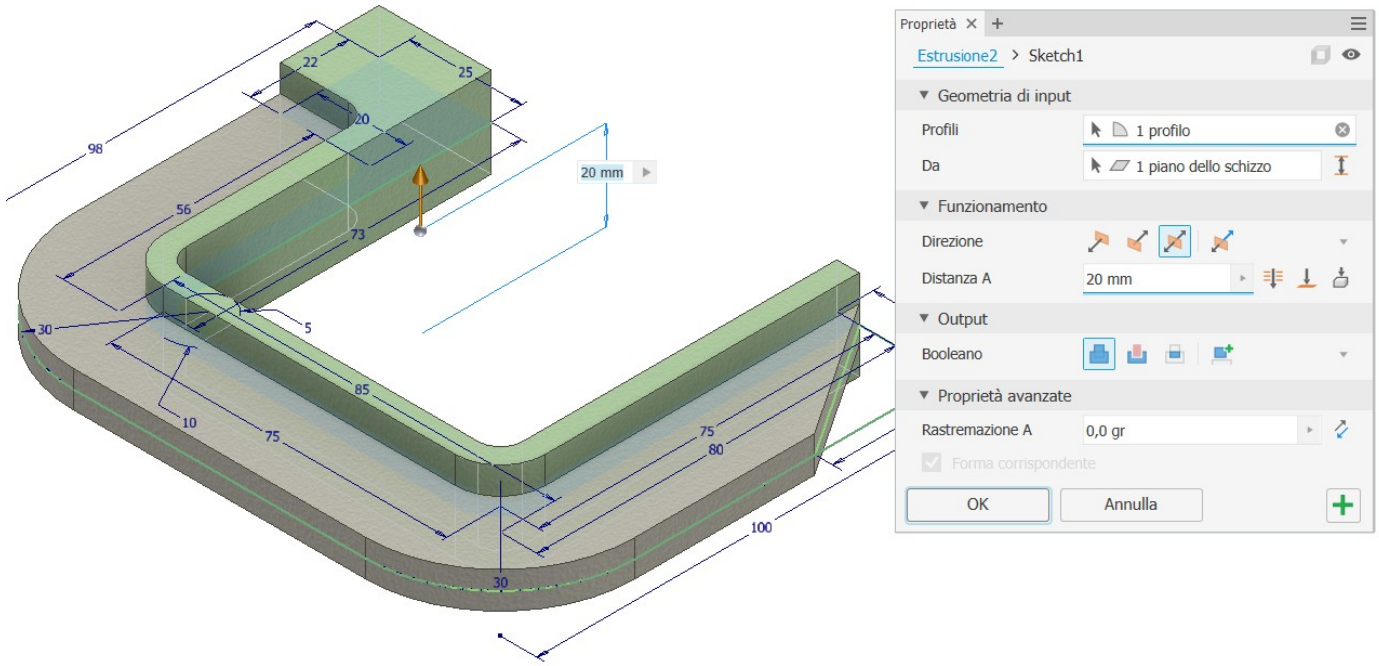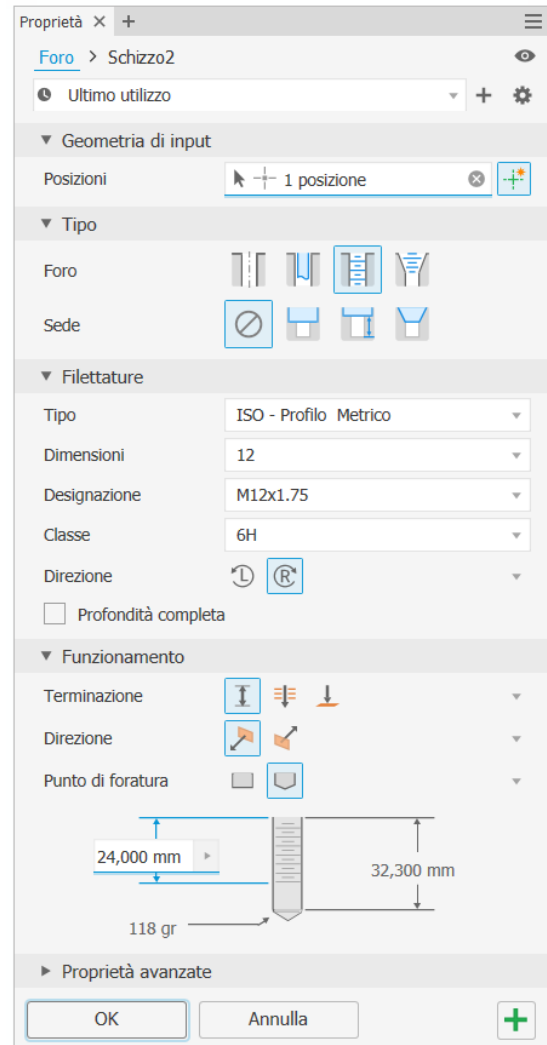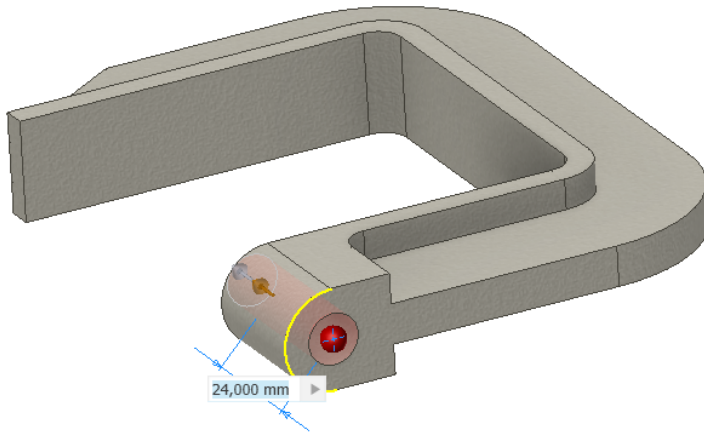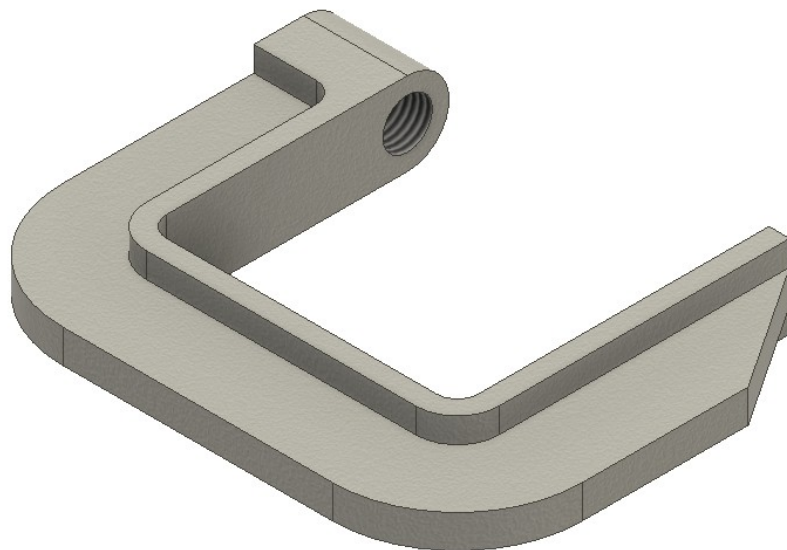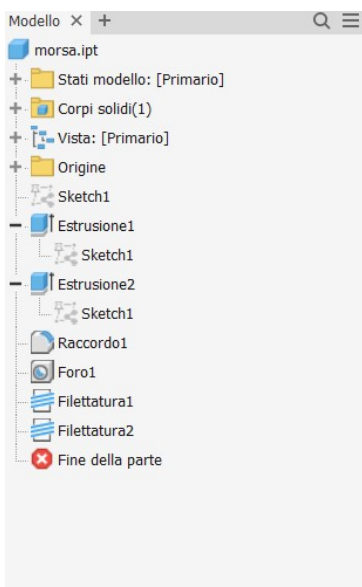

Join the two edges shown in the figure.
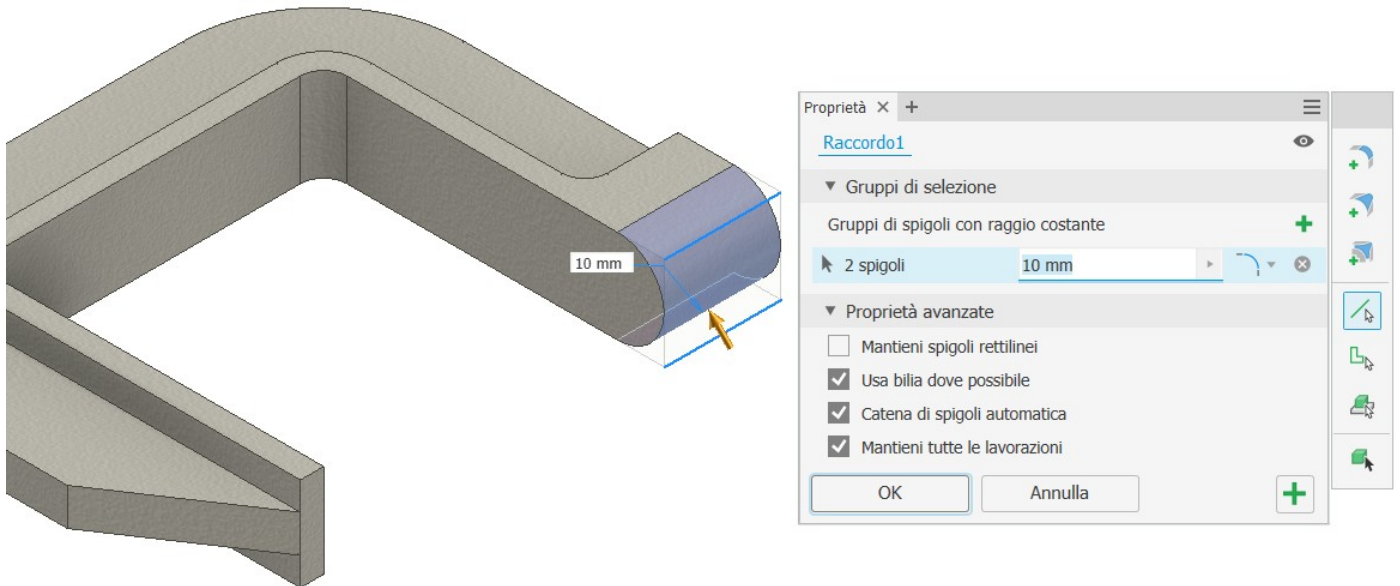
Drill an M12 mm threaded hole on the indicated face by selecting the cylindrical side surface to automatically locate the centre of the hole. Select the end face of the hole and do not select through.

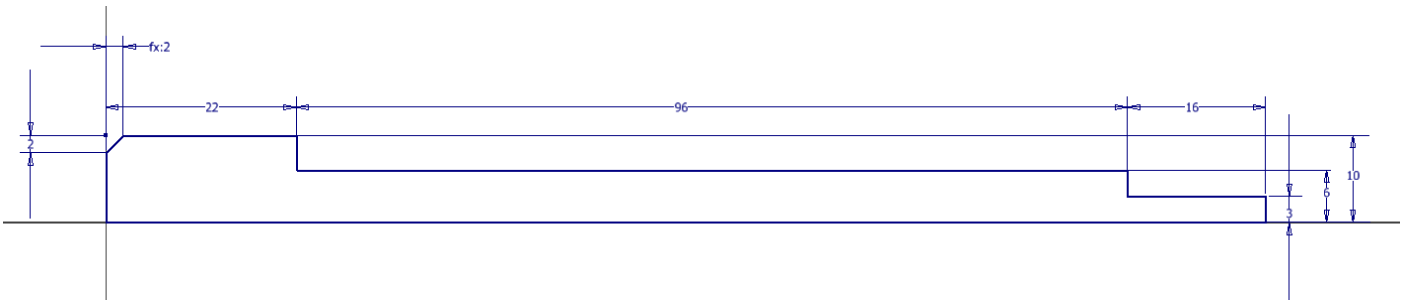If you have difficulty locating the centre of the hole, create a sketch on the face and place a reference 'dot' in the centre.
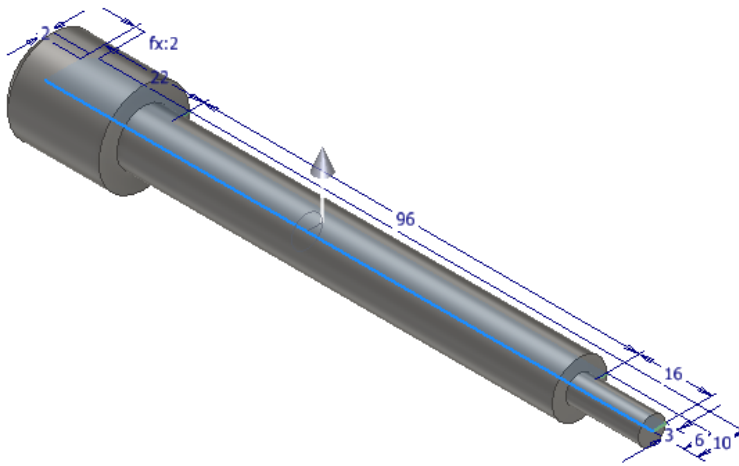

NOTE:

The two extrusions 1 and 2 share the same sketch 1 (not mandatory but convenient in this case).
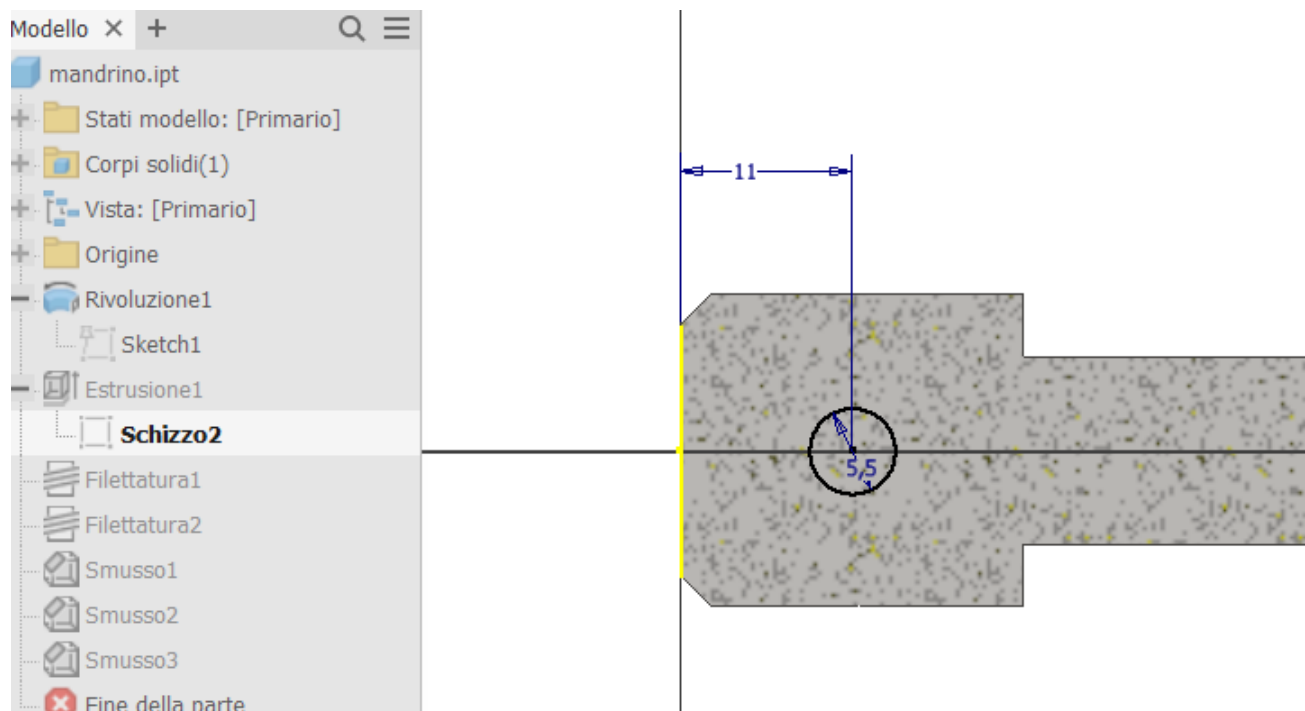
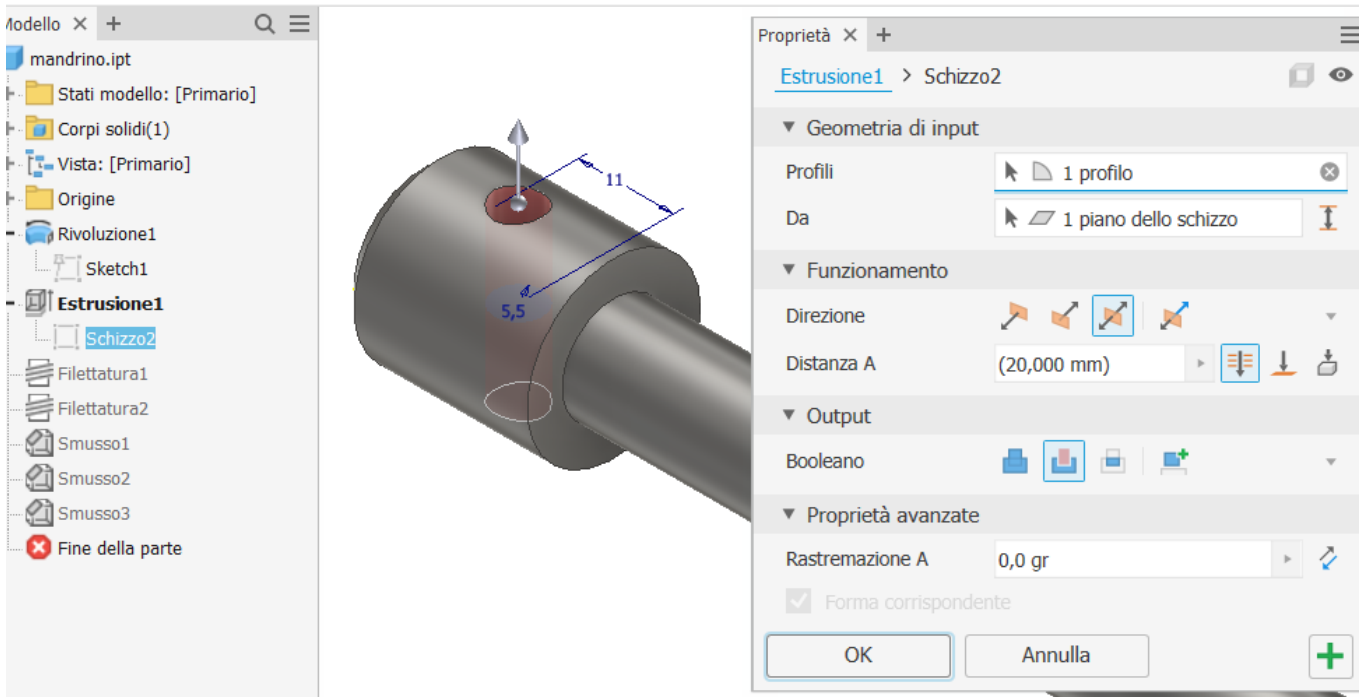Create the following sketch using the axes of the sheet as reference.

Create a revolution solid as in the figure by selecting the profile on the sketch.
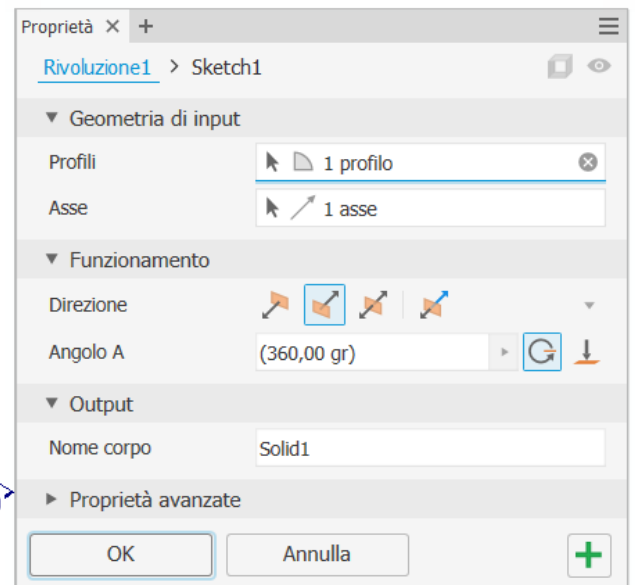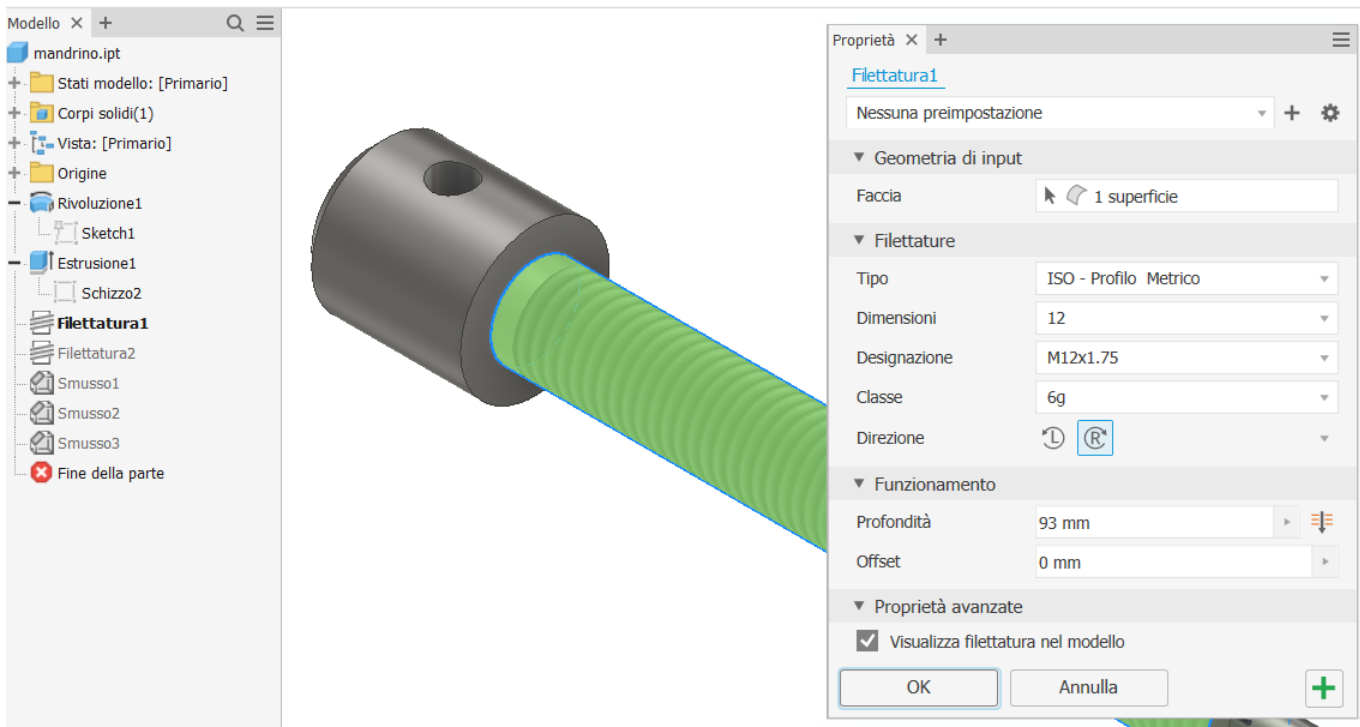

On the horizontal plane of symmetry of the workpiece, draw the through-hole sketch.
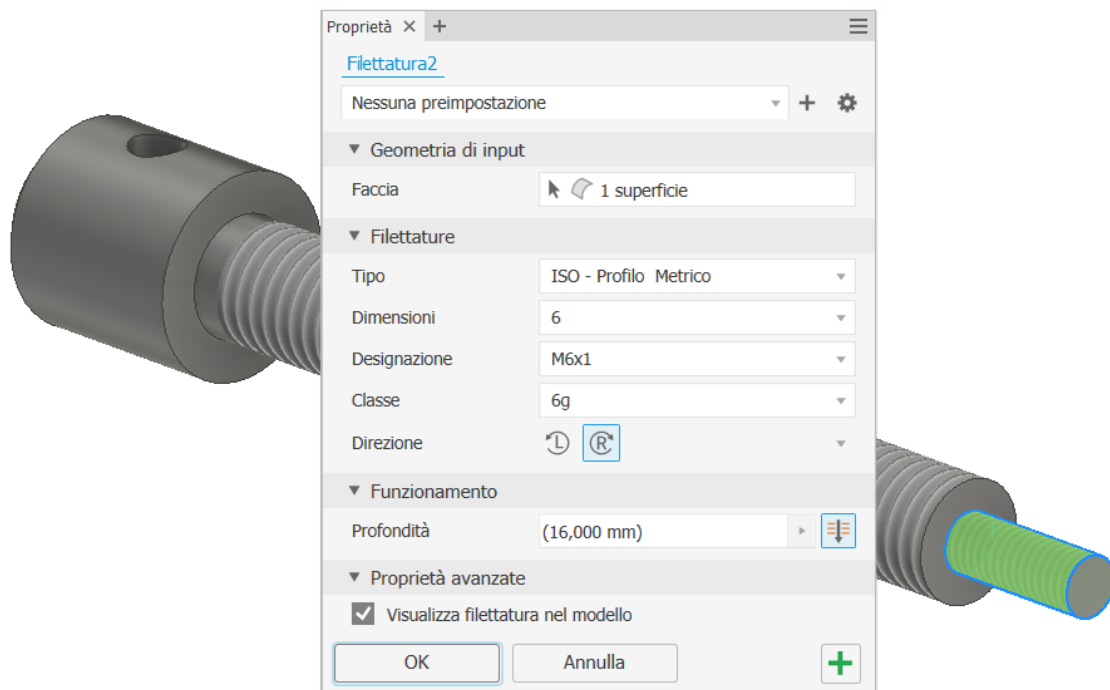
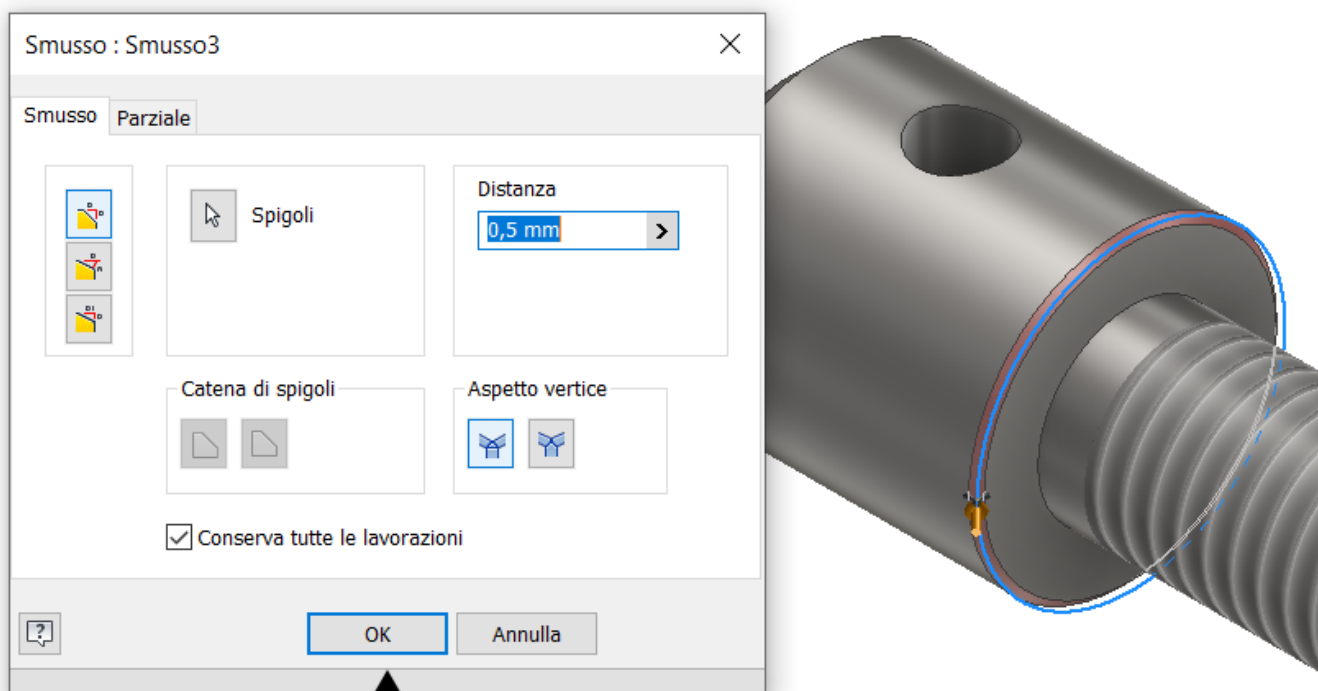Perform a through extrusion in both directions.

Continue with spindle threading.

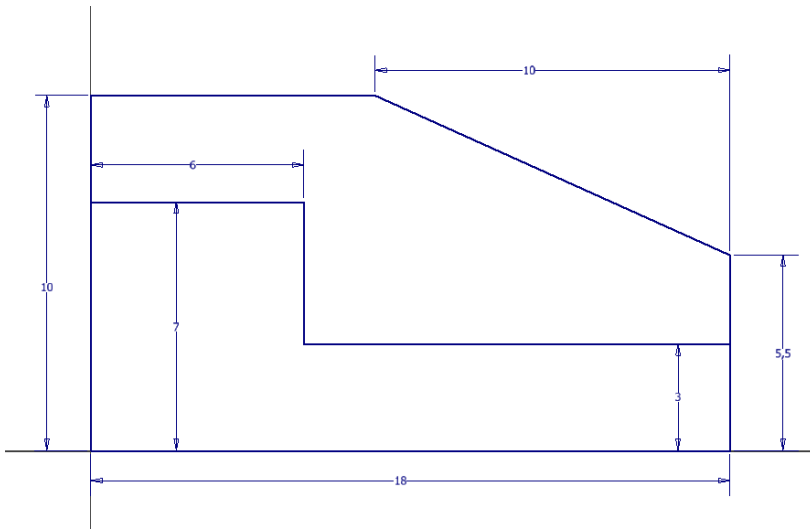Continue with the threading of the end part housing the nut.
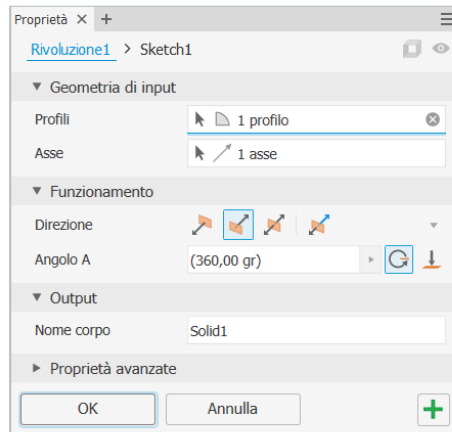
Add chamfers to the workpiece.
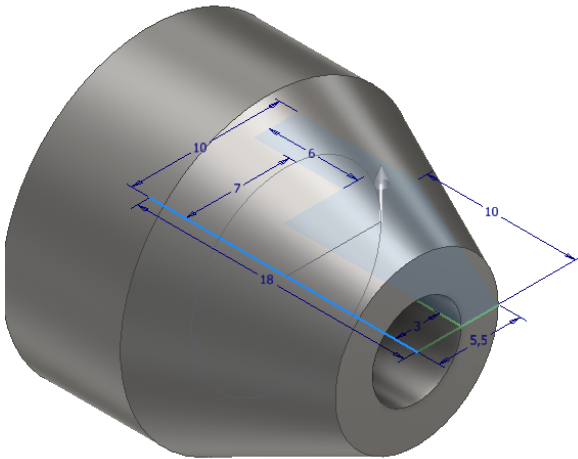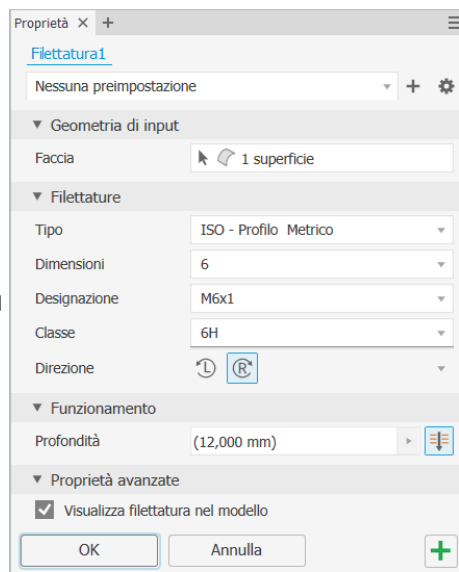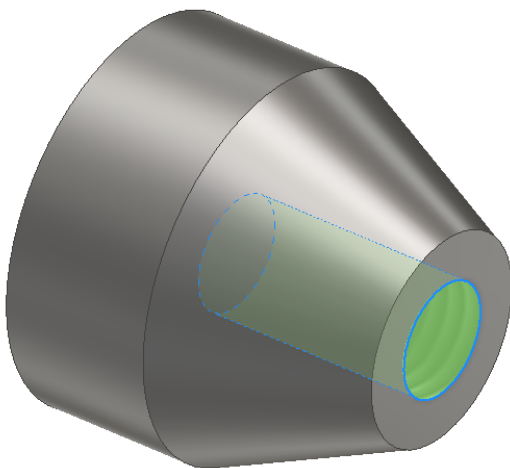
Create the following sketch using the axes of the sheet as reference.

Create a revolution solid as in the figure by selecting the profile on the sketch.

Complete the part with M6 through-hole

CREATION OF 2D VIEWS OF PARTS

In most cases, a workpiece is created using simple machine tools (lathes, mills, drills, etc.) that perform flat machining. It is therefore essential to provide 2D views of the part that allow the operator to perform the required machining.
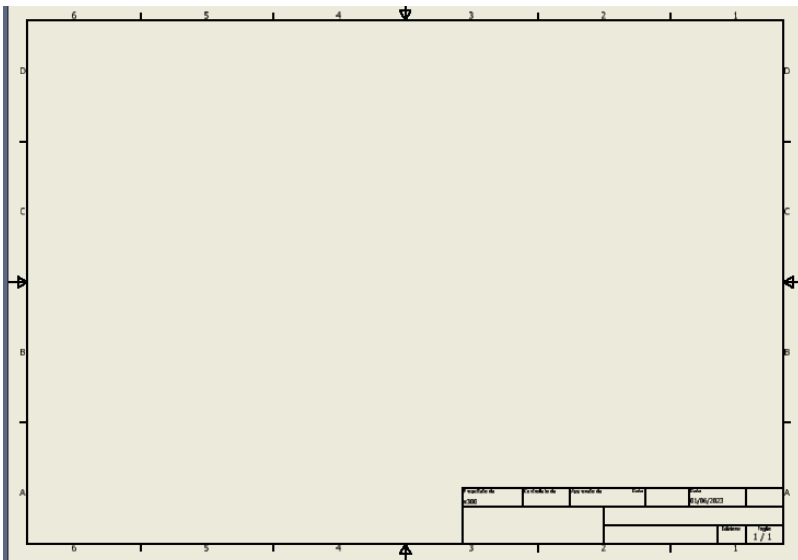
Open the part you wish to draw in 2D.
From the MENU "*file*" "*new*" you reach the mask for creating a 2D drawing.
Select "ISO.dwg" (the extension .dwg identifies Autocad-compatible 2D drawings) and click create.
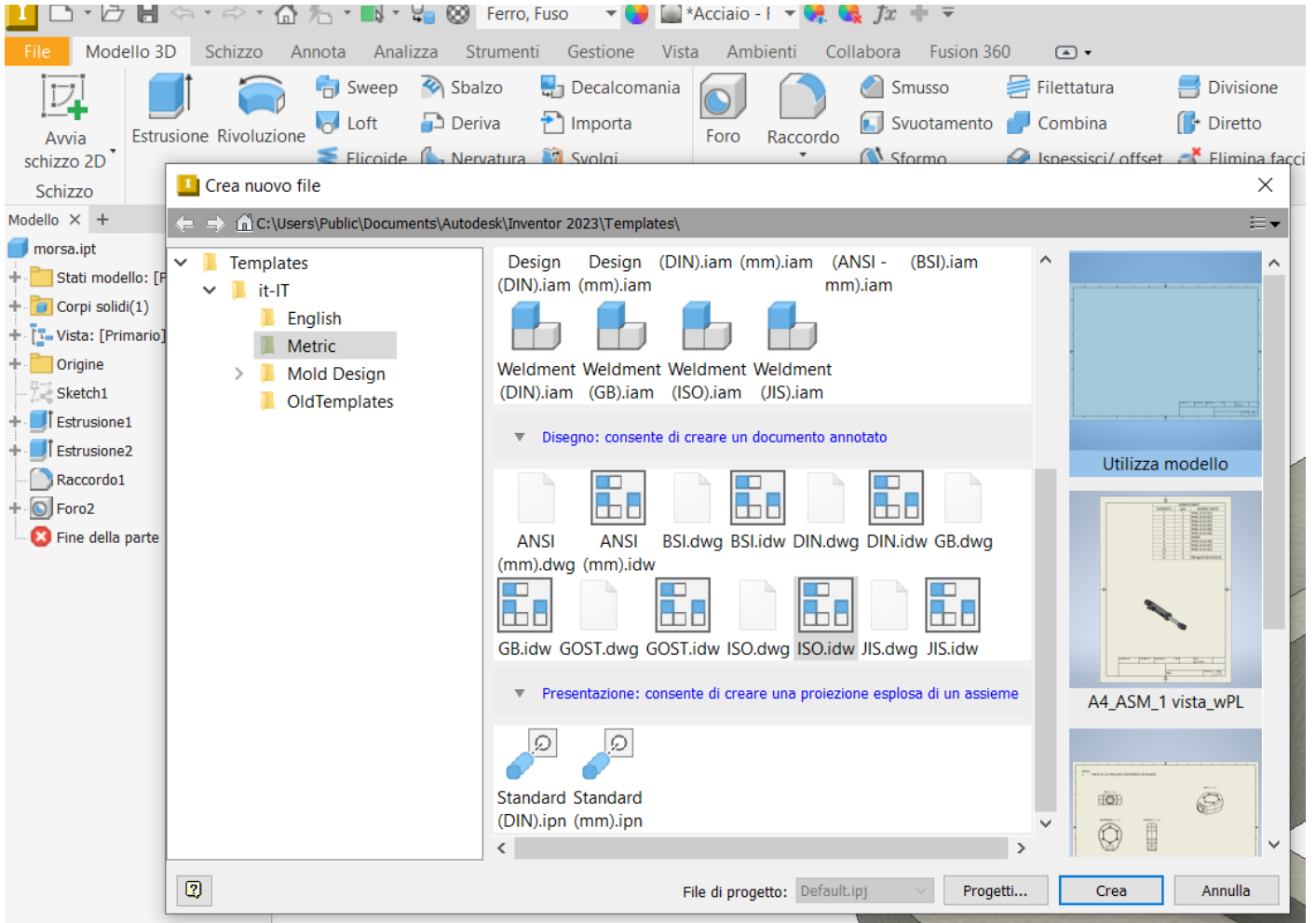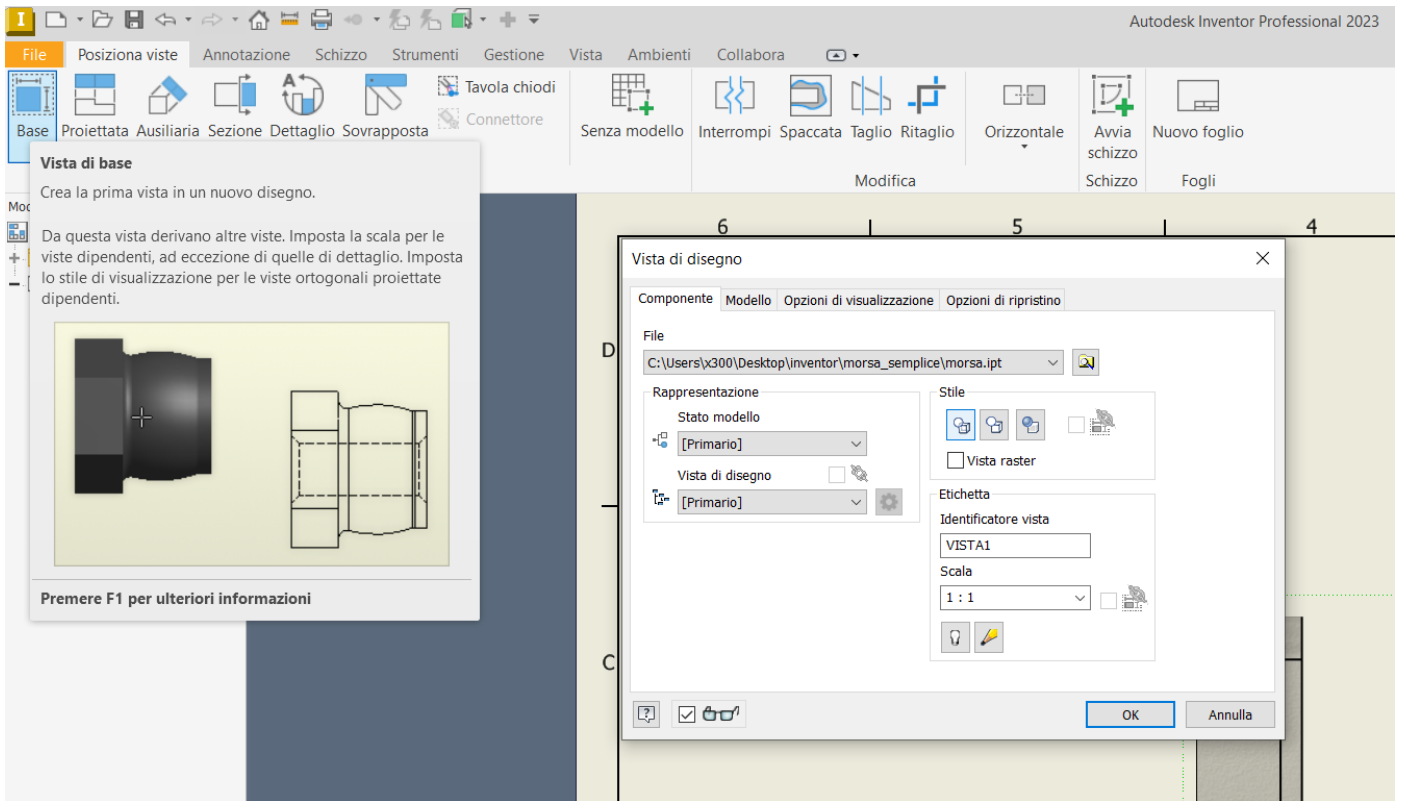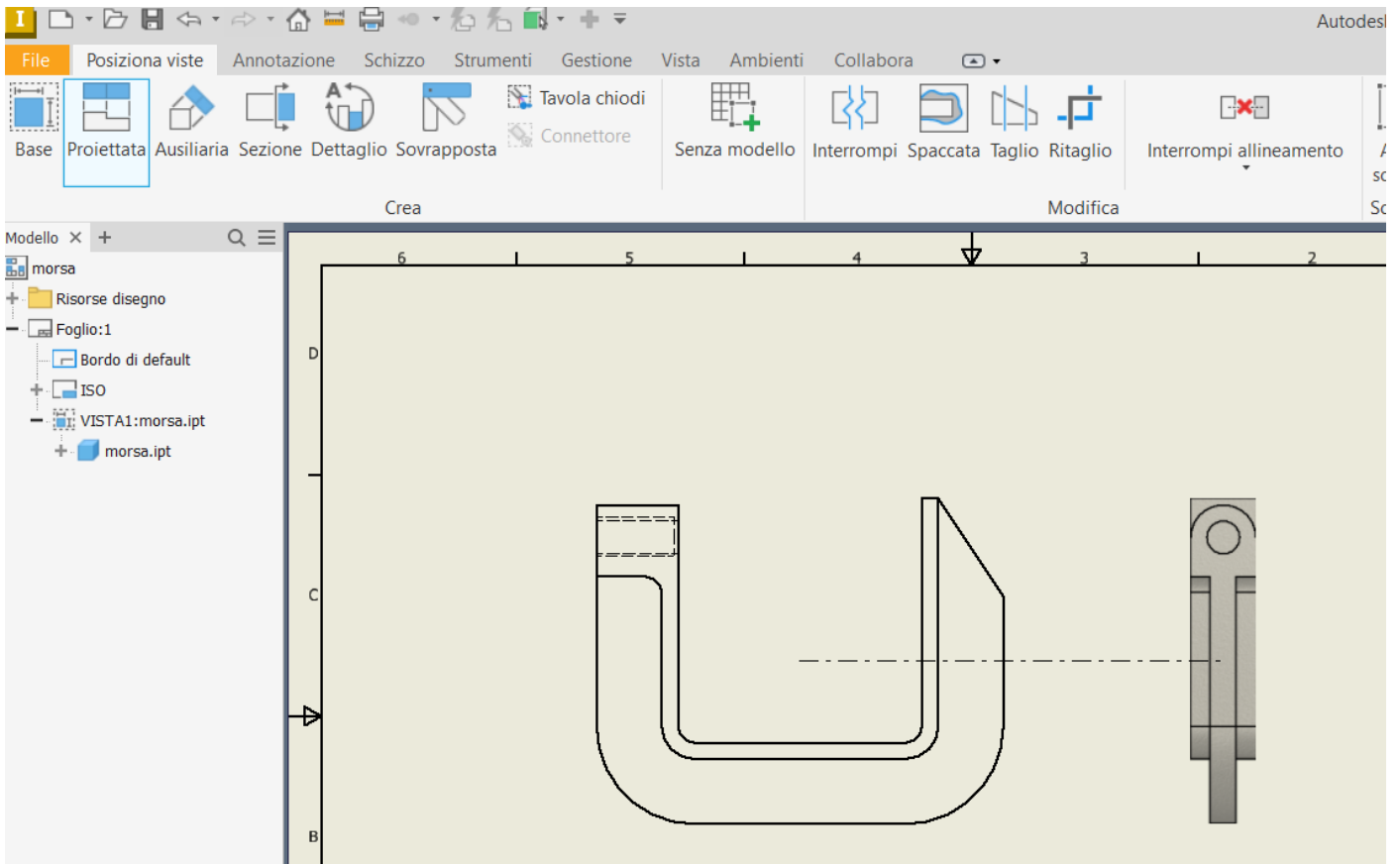

An empty 2D drawing opens.

2D DRAWING VICE

Click on 'BASIS' to locate the main view of the workpiece and indicate the desired scale.
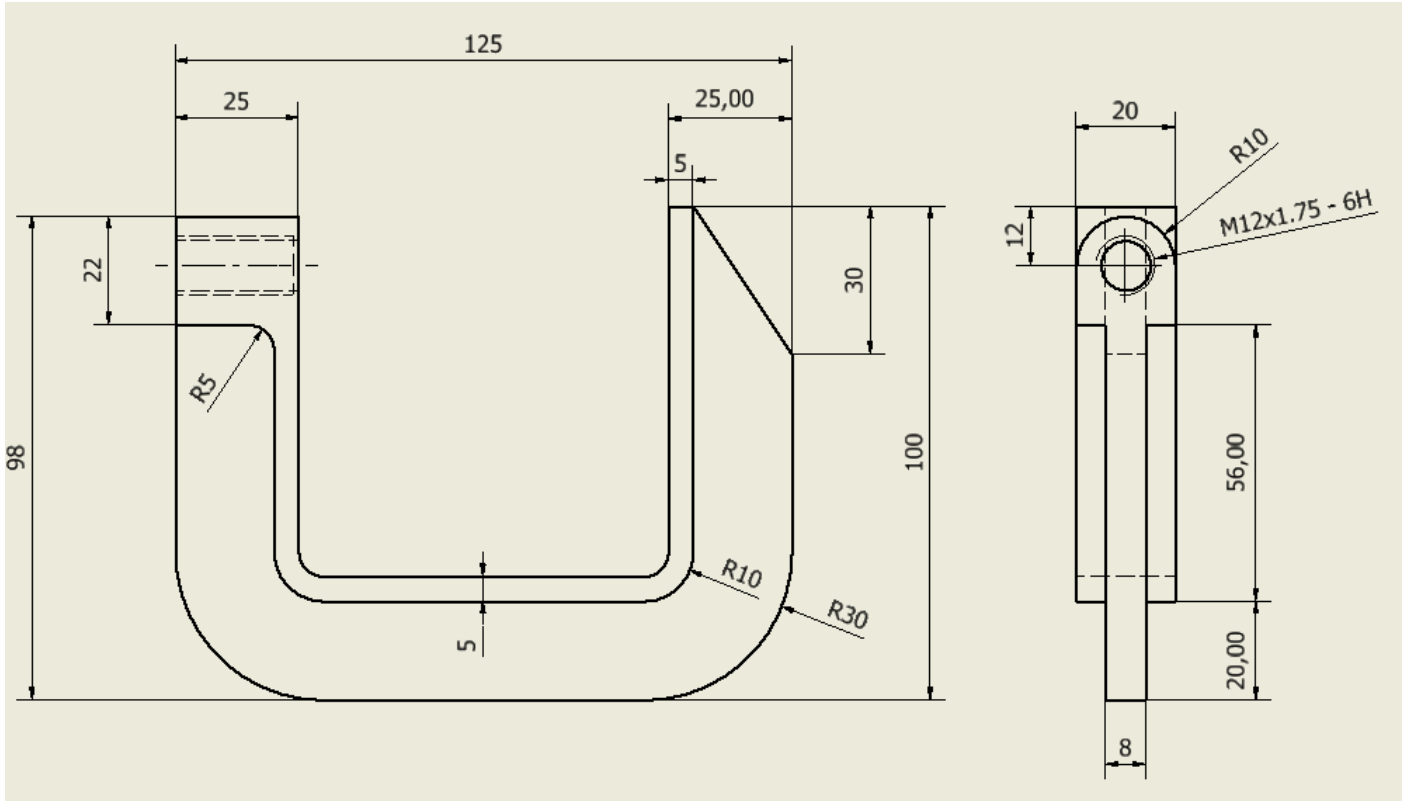

Click on 'Projected' to add the secondary views required to fully describe the part.
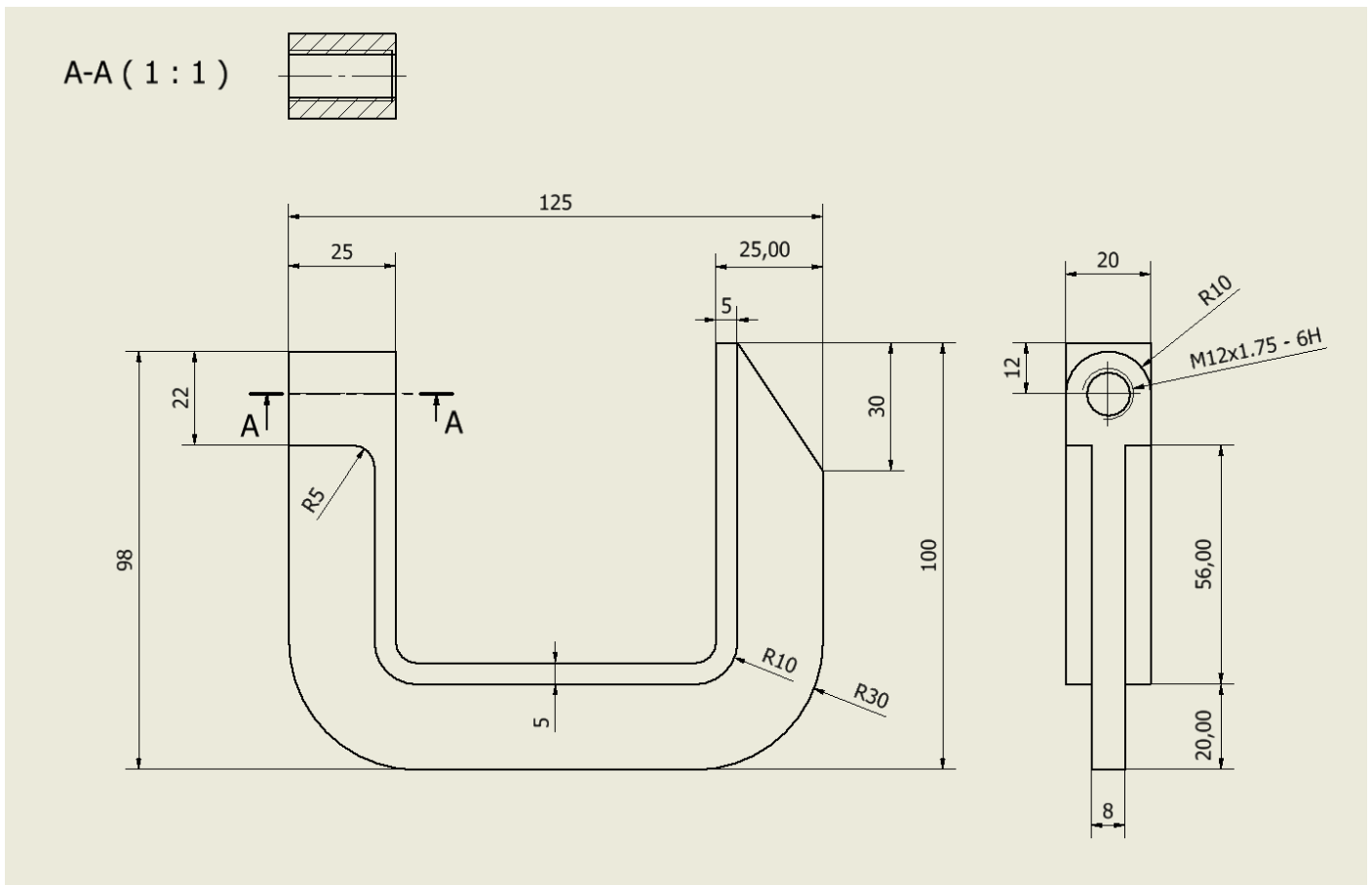
The drawing must be completed with axes, hole centres, dimensioning, etc. in accordance with the technical regulations.
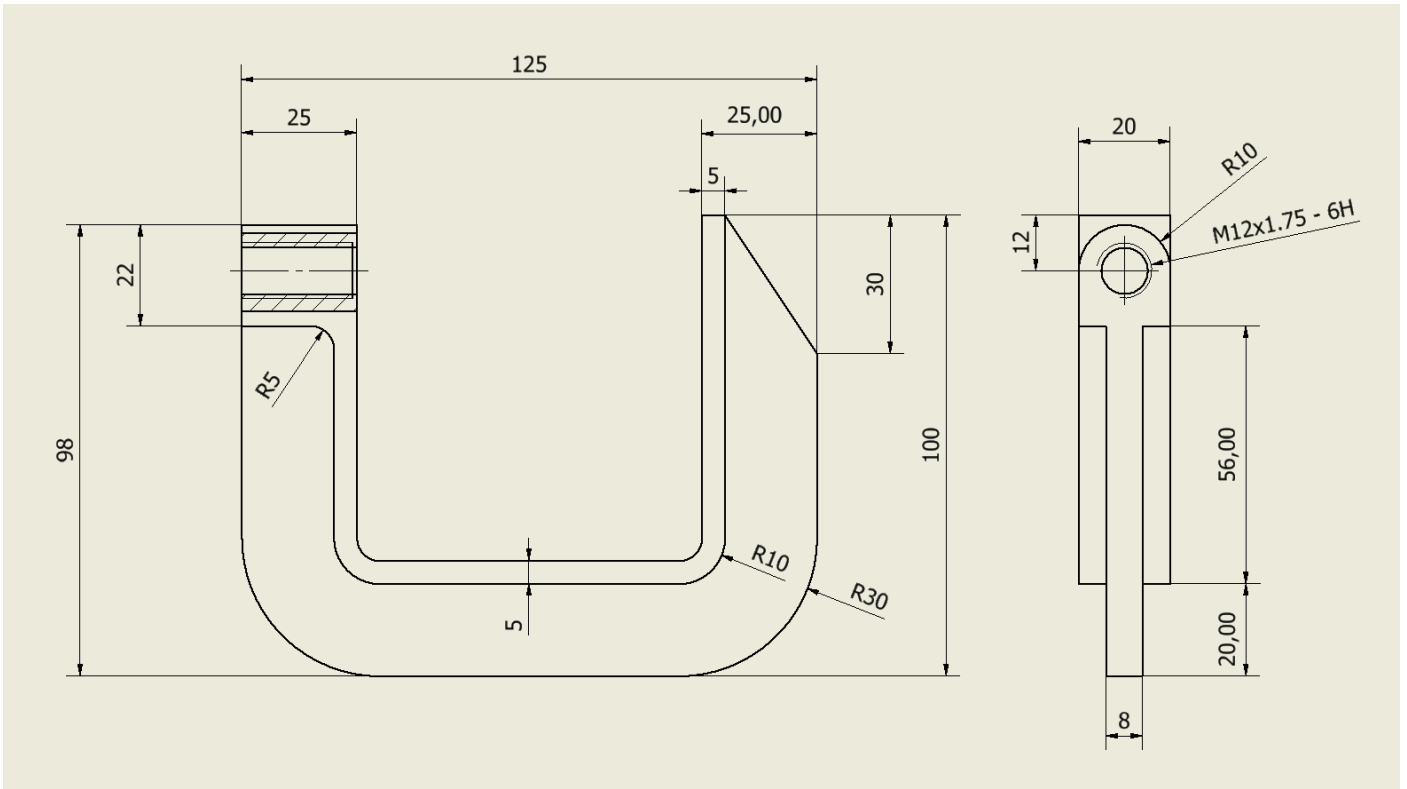
With hidden edges.

With section to highlight through-hole
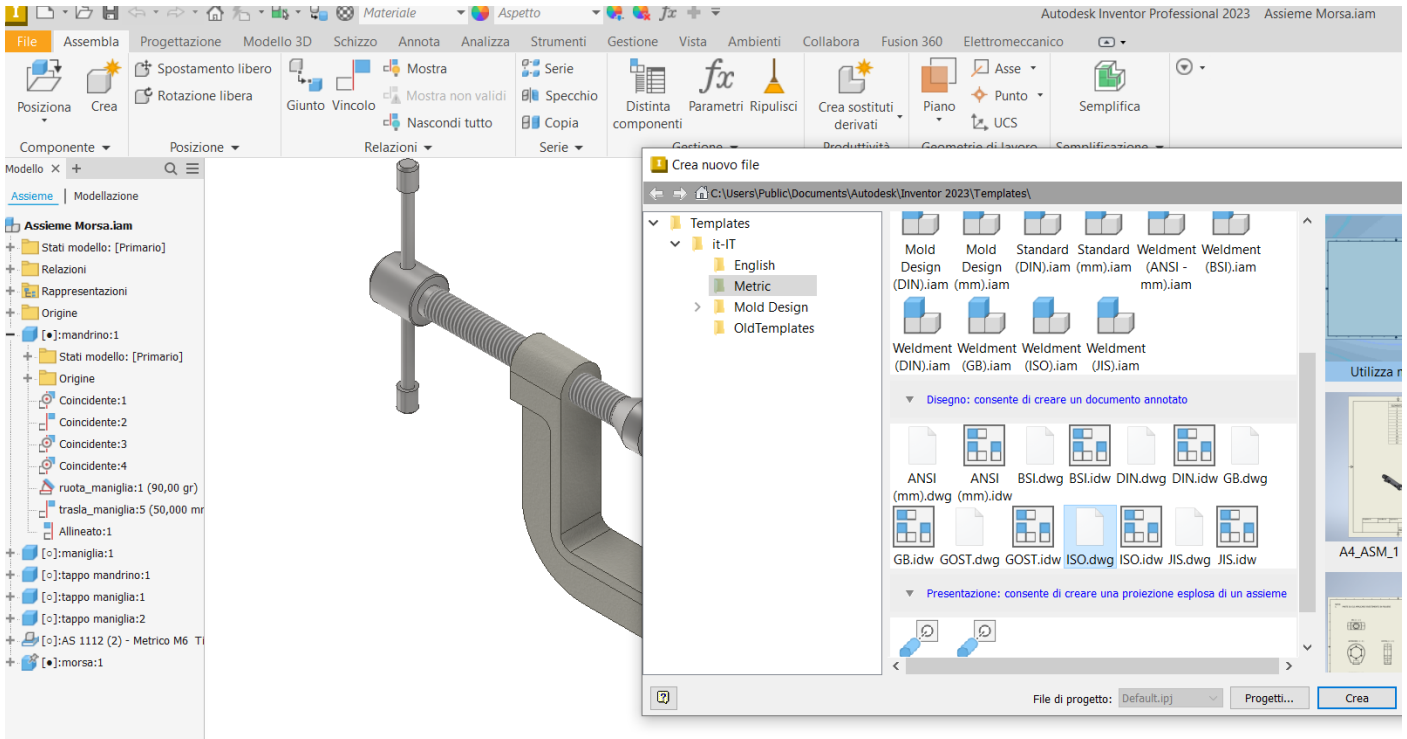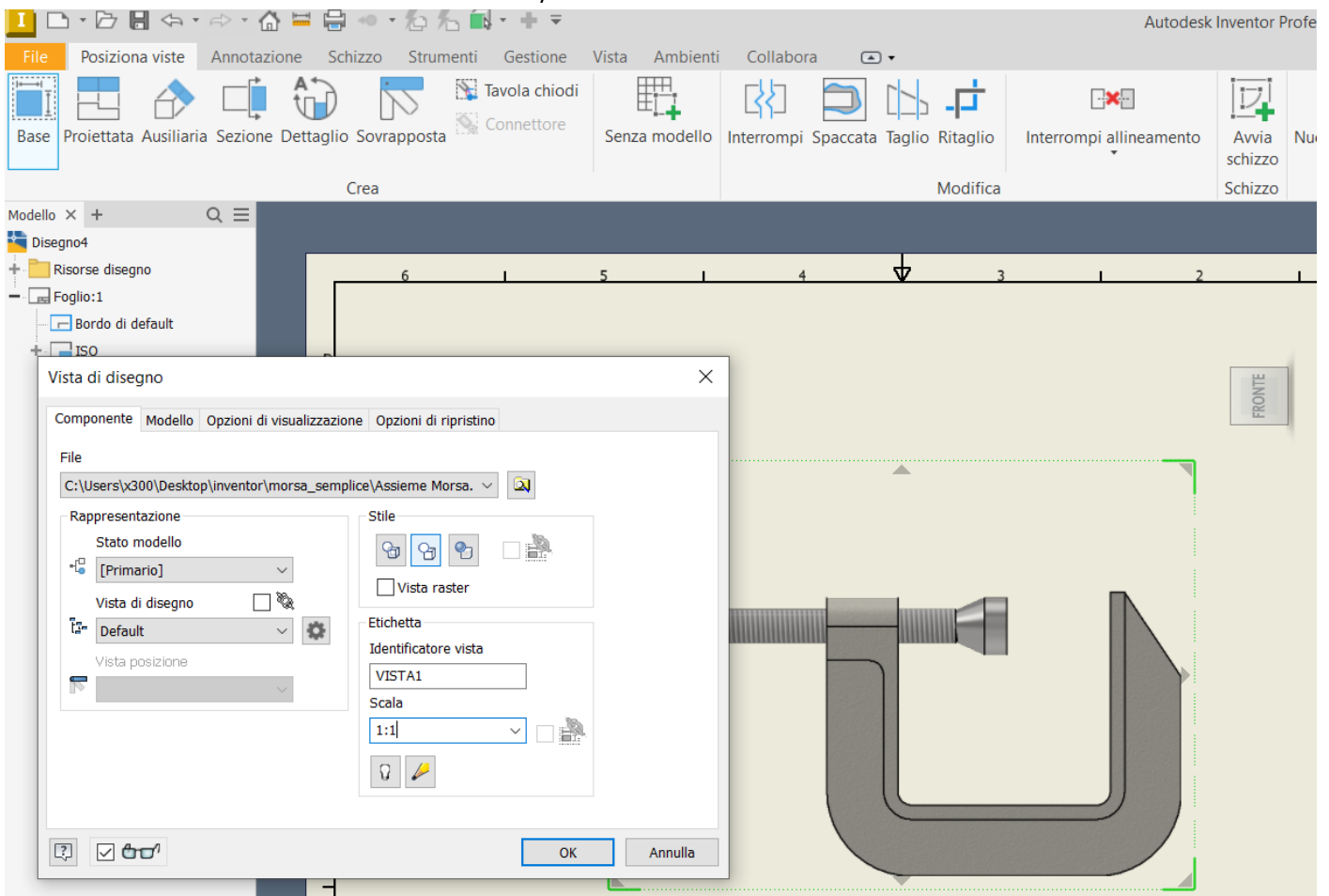
With split view at the through-hole.

Open the assembly you wish to draw in 2D.

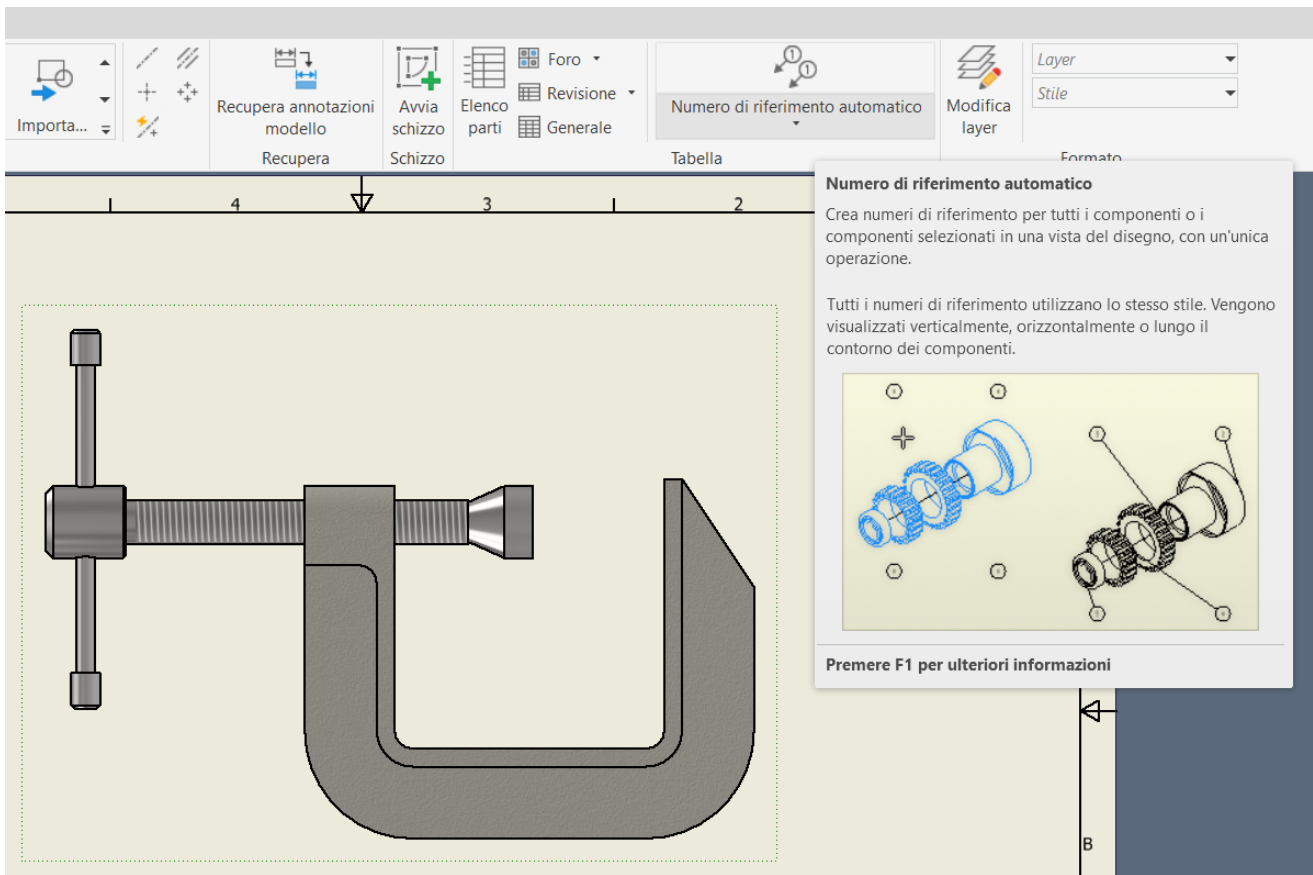From the MENU "*file*" "*new*" you will reach the mask for creating a 2D drawing.

Select "ISO.dwg" (the .dwg extension identifies Autocad-compatible 2D drawings).
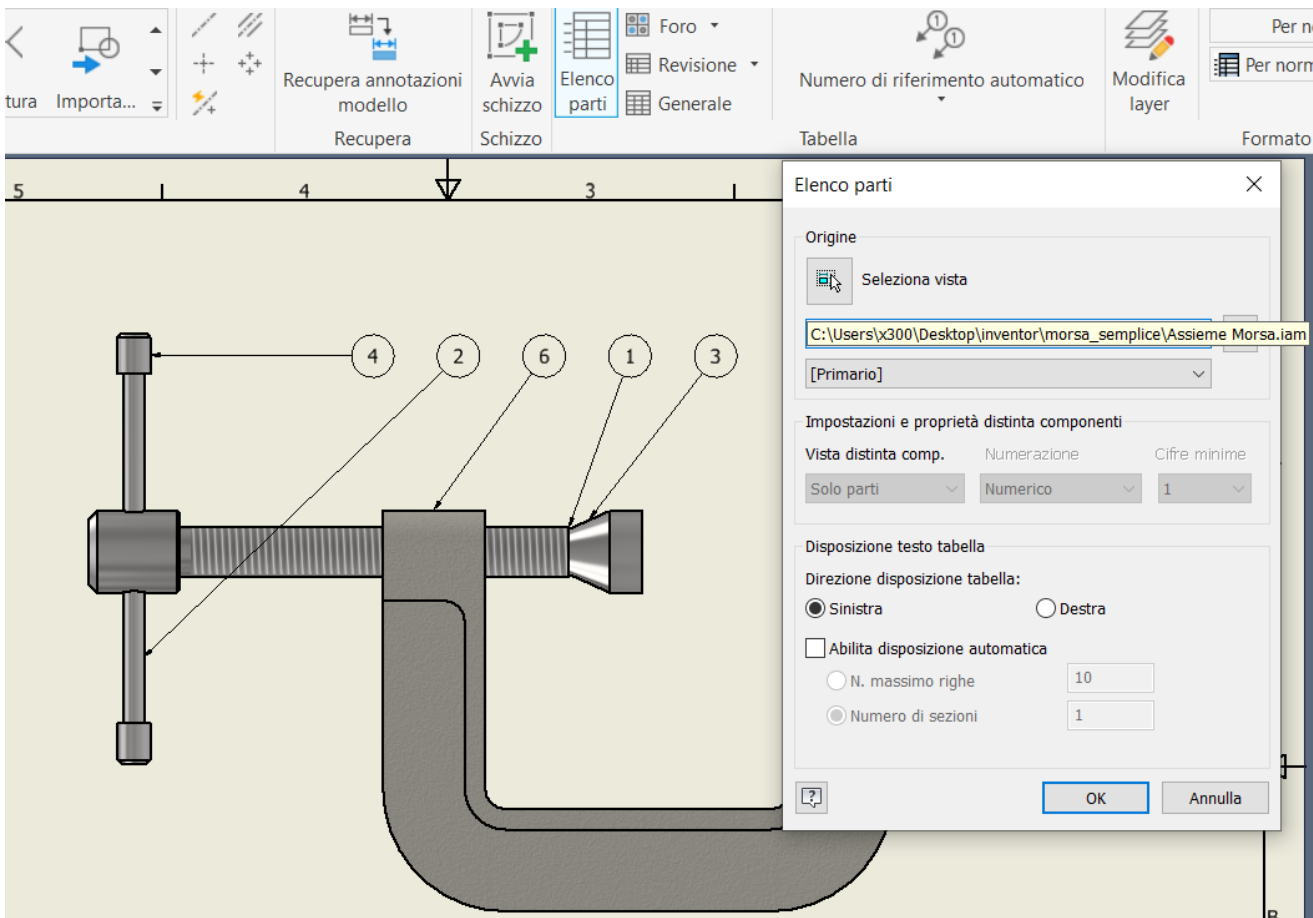

Click on BASE to locate the main view of the assembly and indicate the desired scale.
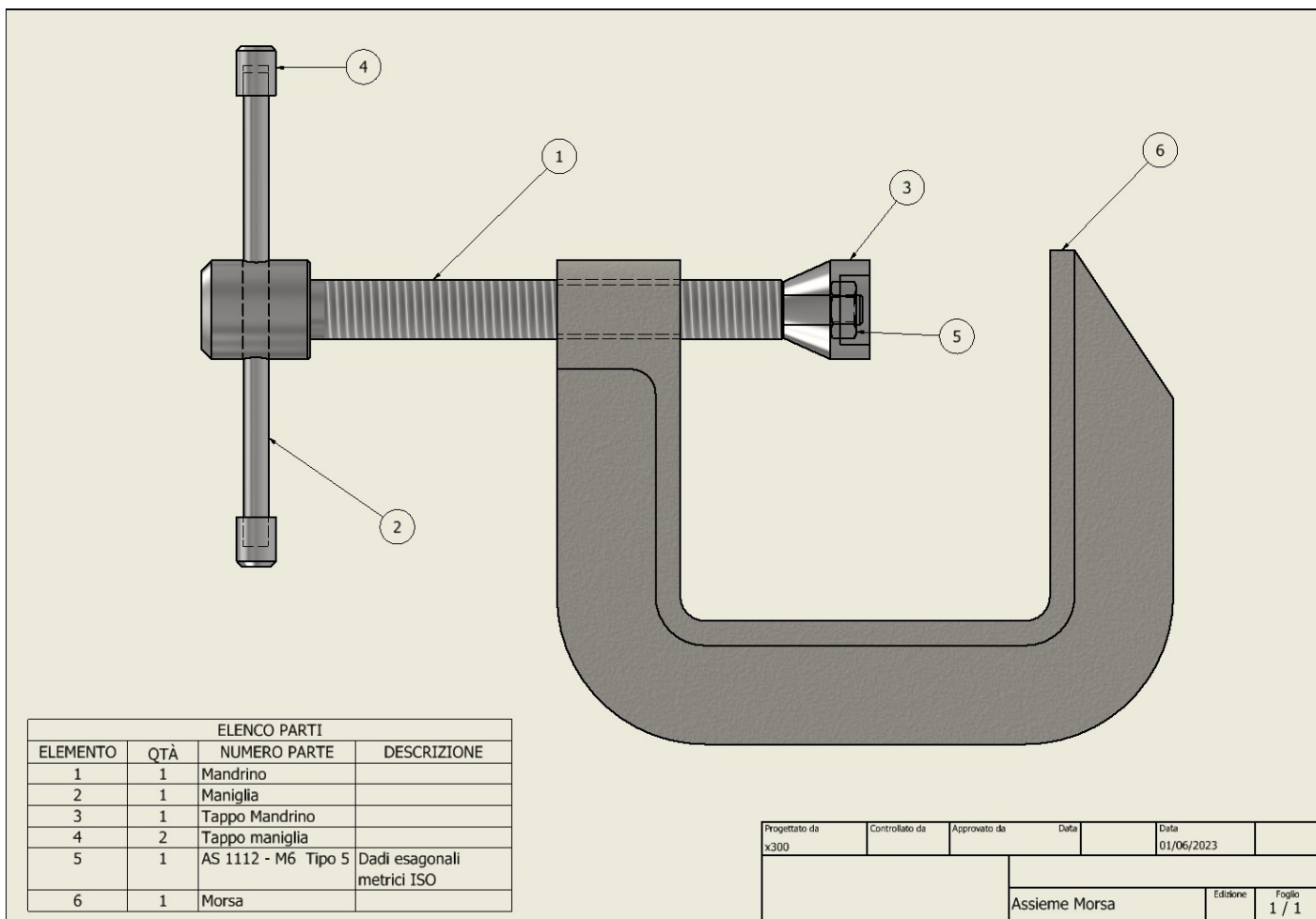
Use 'automatic reference number' to enter references to individual parts.

Use 'parts list' to insert the table with the parts.

Drawing together final parts.

ELENCO PARTI			
ELEMENTO	QTÀ	NUMERO PARTE	DESCRIZIONE
1	1	Mandrino	
2	1	Maniglia	
3	1	Tappo Mandrino	
4	2	Tappo maniglia	
5	1	AS 1112 - M6 Tipo 5	Dadi esagonali metrici ISO
6	1	Morsa	

| Progettato da x300	Controllato da	Approvato da	Data	Data 01/06/2023	
		Assieme Morsa		Edizione	Foglio 1 / 1

From the "file" MENU "*new*" you will arrive at the assembly creation mask.

Select "standard (mm).iam" (the .iam extension identifies an assembly).
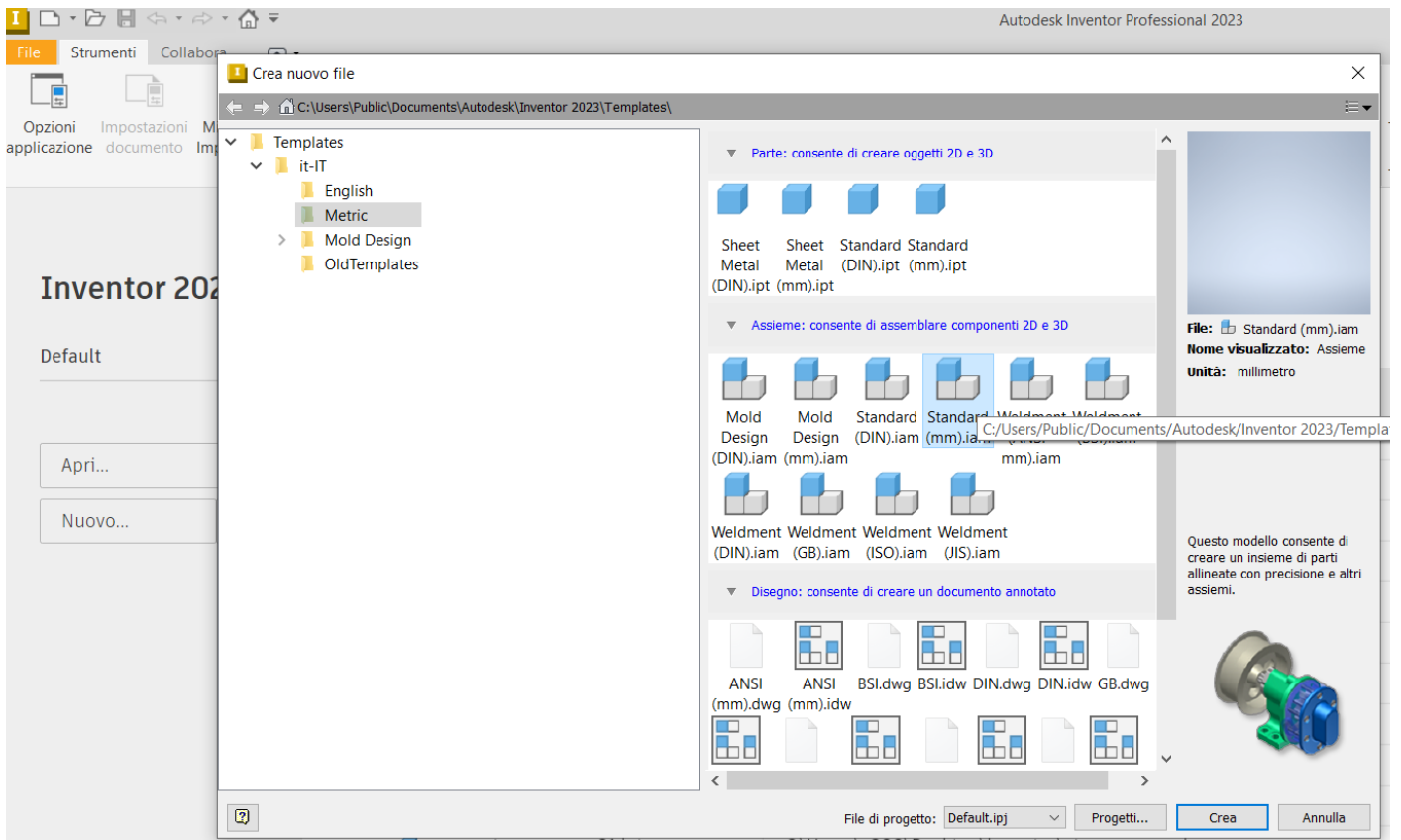
Immediately after confirming with 'create' you must save the file (format .iam) with the name of the structure/machine you are creating in the project folder containing all individual parts.

The folder will have a structure similar to the one below:
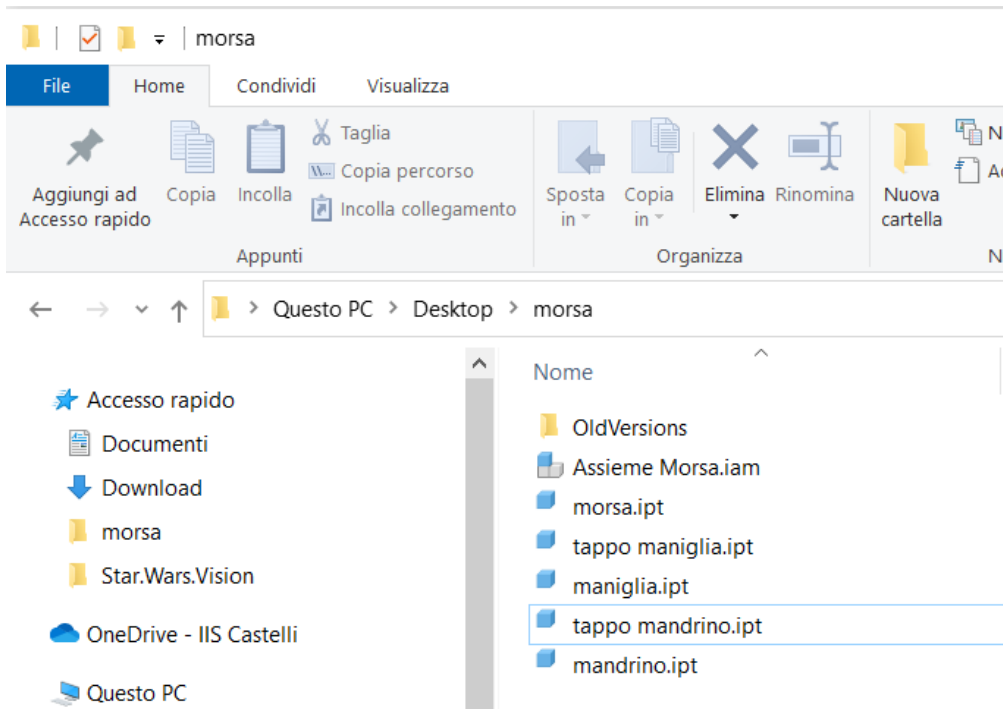
The assembly just created with the procedure described will be an empty 3D document.
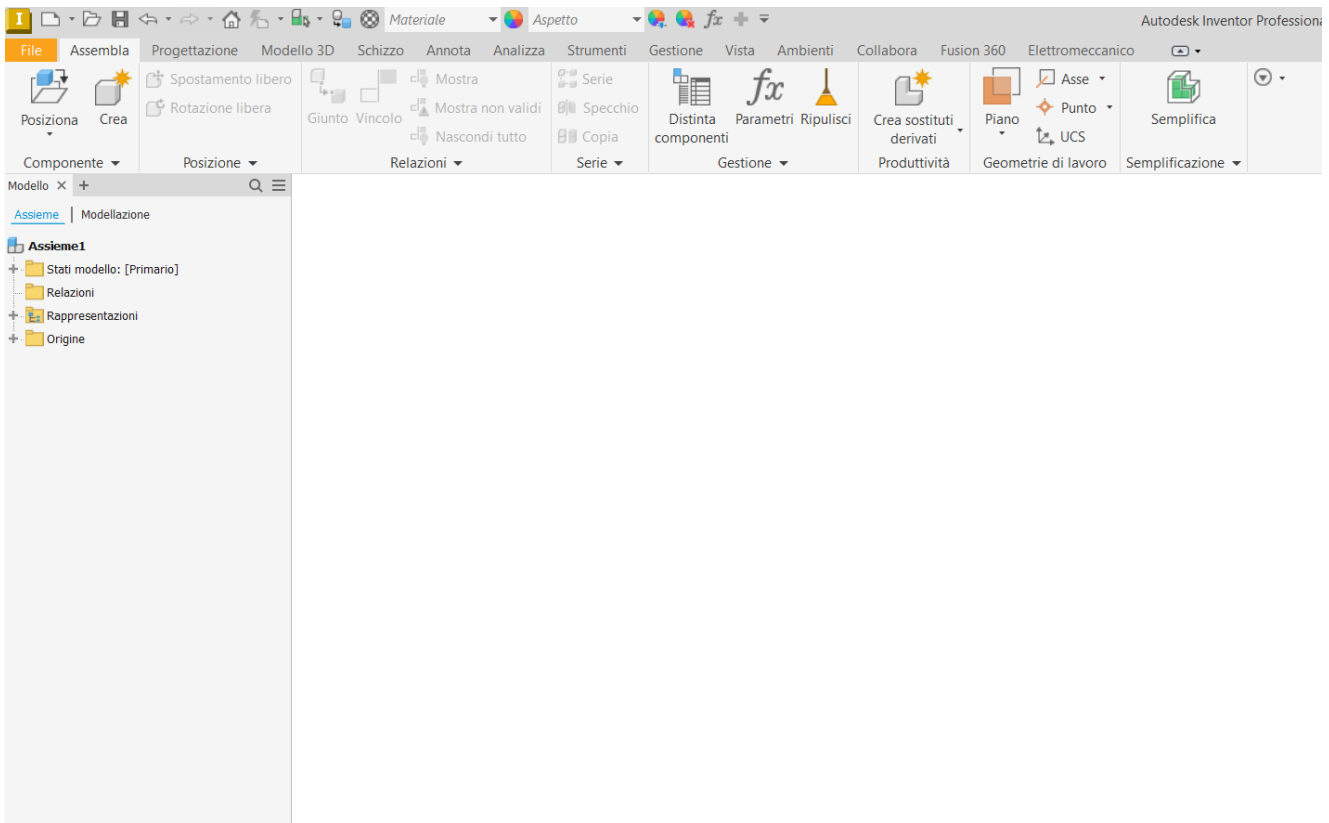

The first thing to do at this point is to insert the main part of the structure to be assembled into the assembly.
Around this part, all the other parts must be connected with appropriate "**constraints**" that limit their movement.
A **constraint** is a **geometric constraint** that removes a degree of freedom (movement) from the connected part.
 From the "position" MENU, go to the project folder and select the part (e.g. vice.ipt)
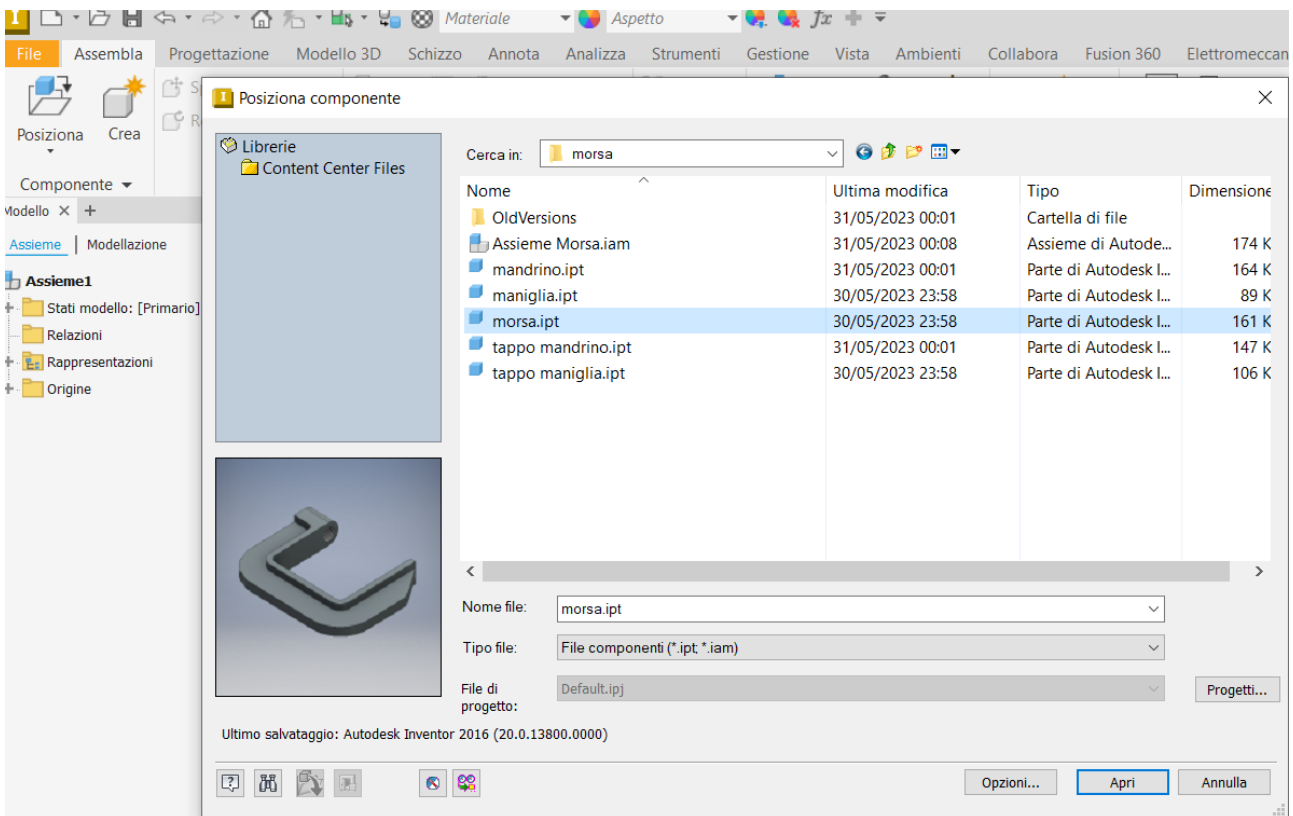
After clicking on 'open' with the right mouse button, click to highlight the '***position fixed in origin***' menu so that the part is positioned and fixed with respect to the axes of the assembly.
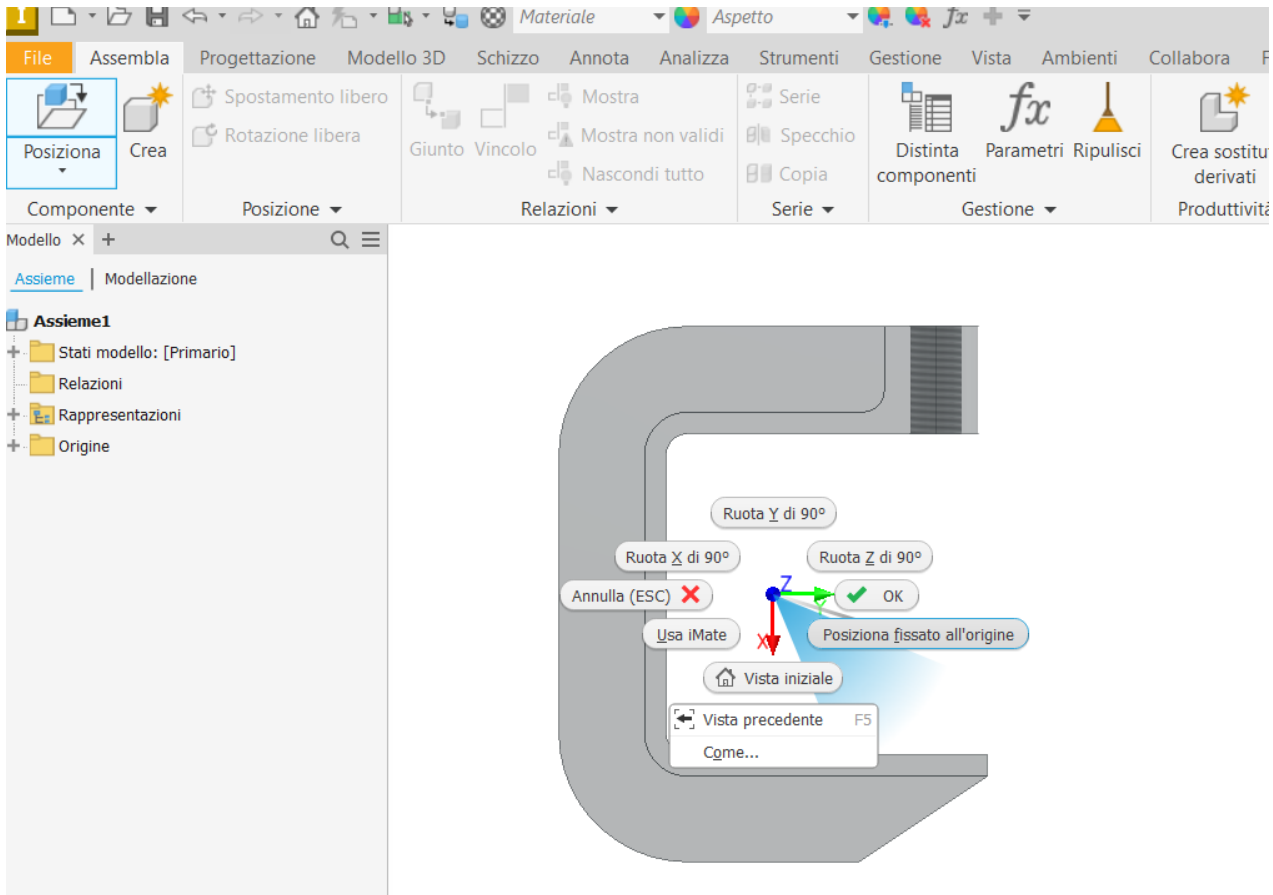

At this point we proceed by positioning all the parts that make up the structure in the assembly, obtaining something similar to that shown in the figure below.
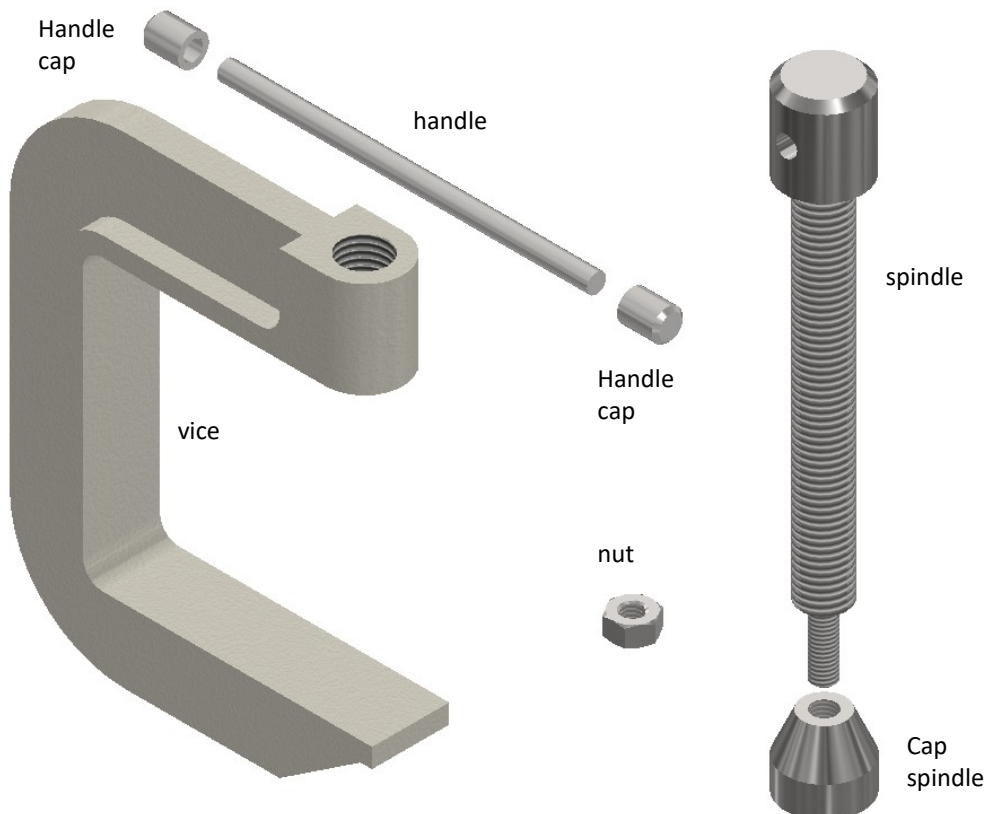
A *constraint* is a *geometric constriction* that removes one degree of freedom (movement) from the constrained part.
A *joint is* a geometric *constraint* that takes away several degrees of freedom (movement) from the constrained part.
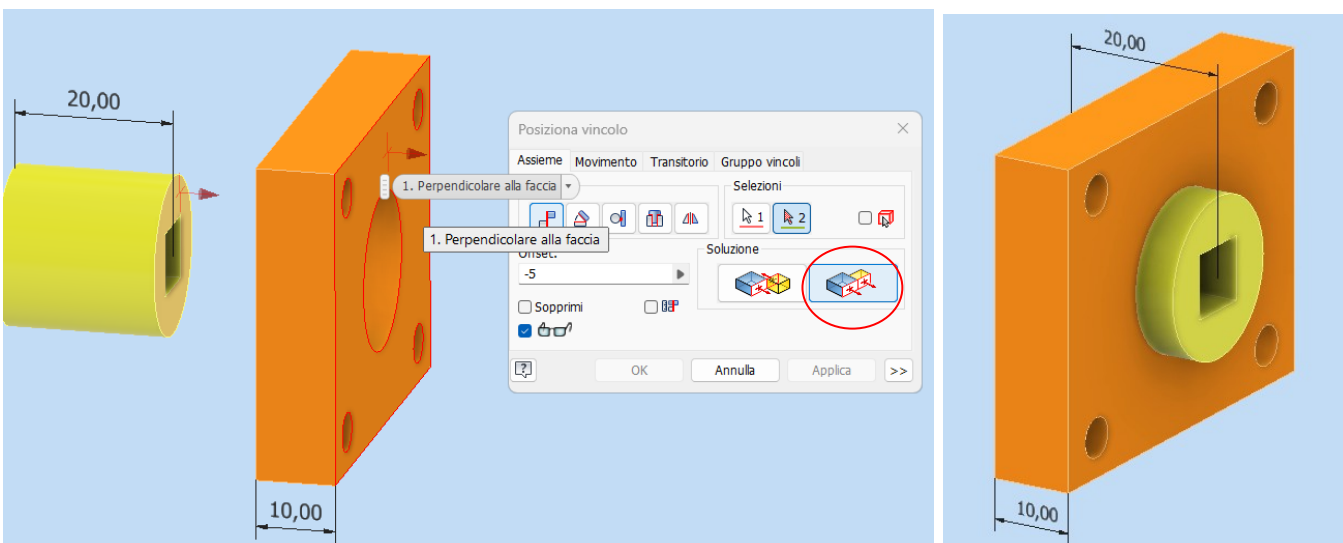In general, if you also have to perform a *KINEMATIC* animation, it is preferable to use simple constraints.
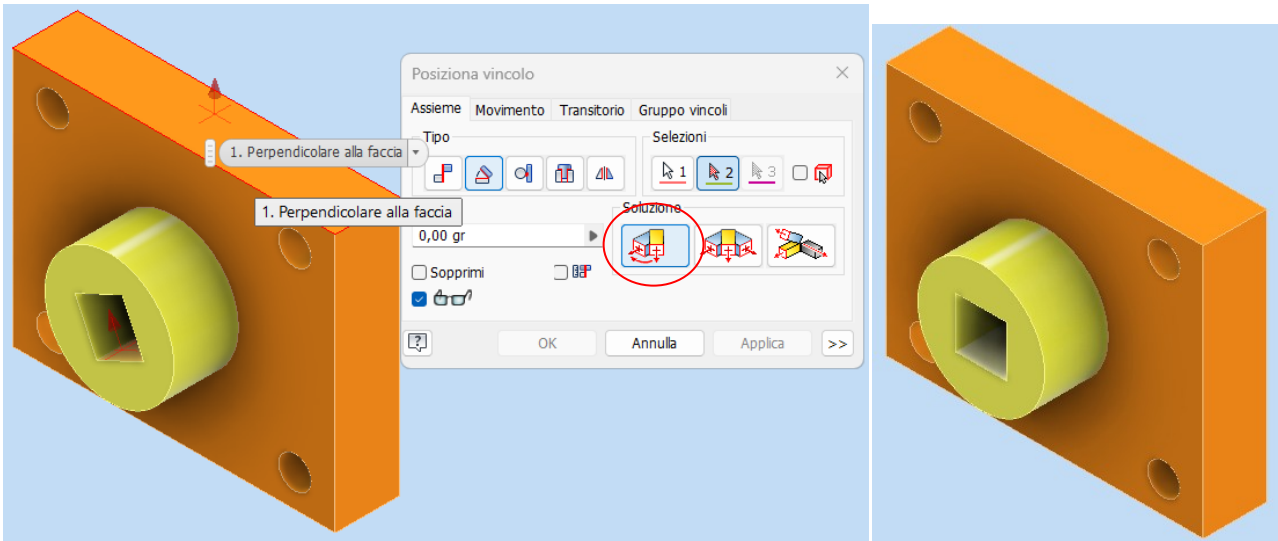
Axial coincidence constraint
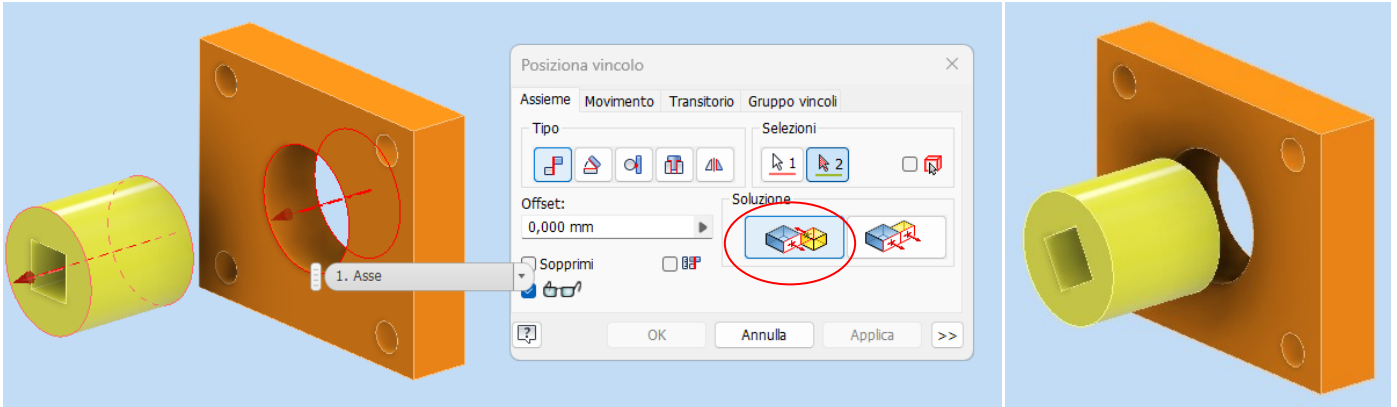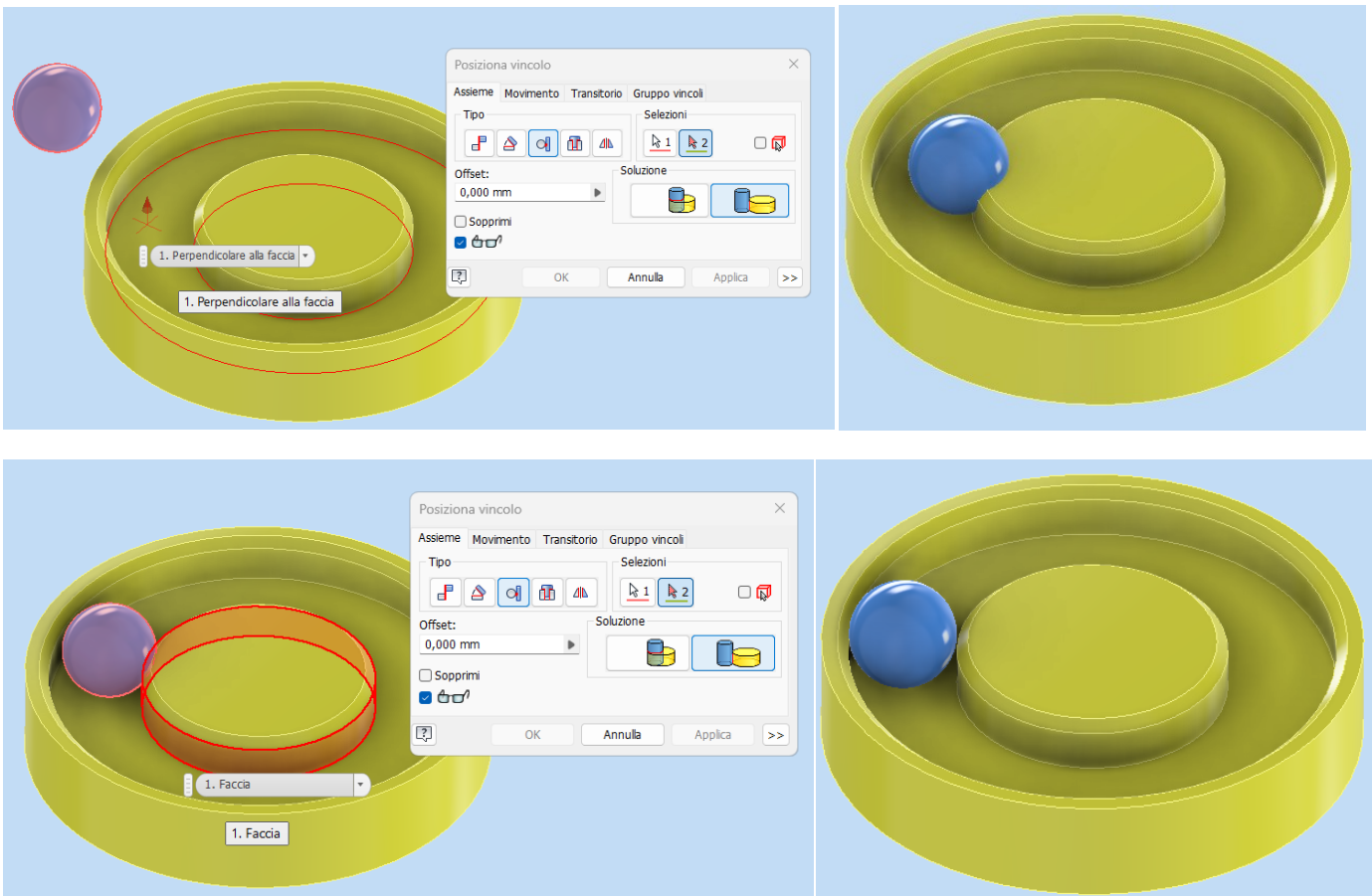

Surface coincidence constraint
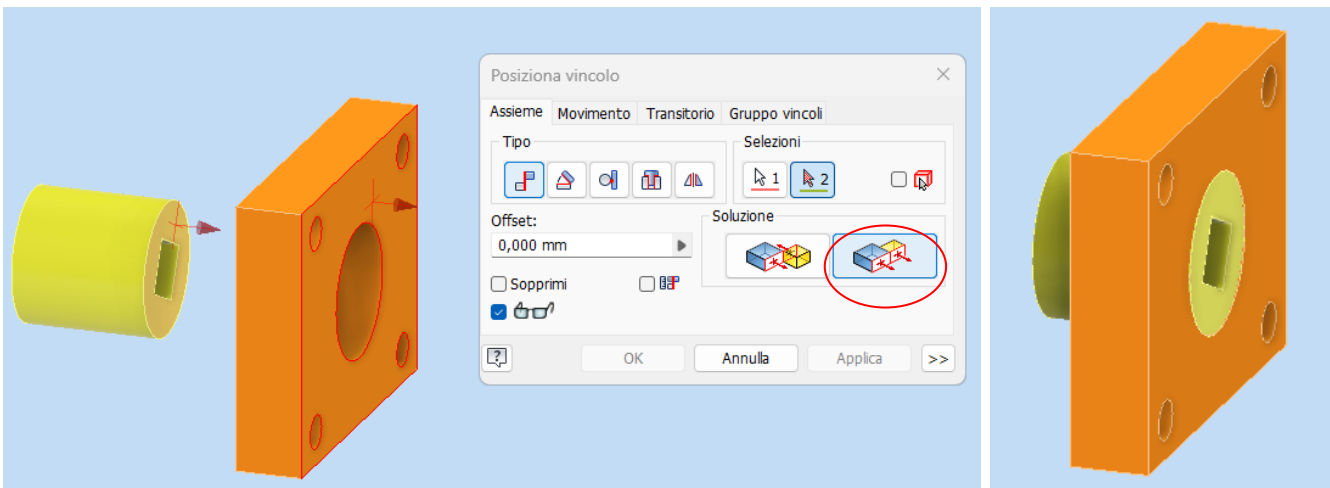
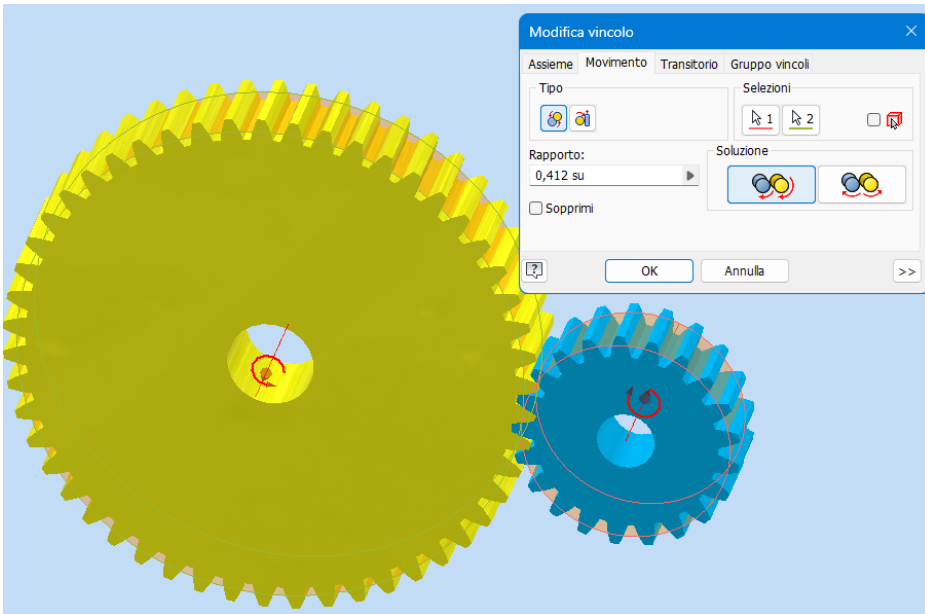**Surface coincidence constraint with offset**

Constraint of parallelism between plane surfaces

**Surface tangency constraint**

## Motion constraint (gear wheels)

Select the primitive diameters of the two gear wheels (orange surfaces) to obtain the gear ratio automatically.

NB:
The initial positioning of the wheel teeth must be done by hand; please note that the model of the teeth generated by Inventor is approximate and therefore it appears that the teeth interpenetrate each other in places.

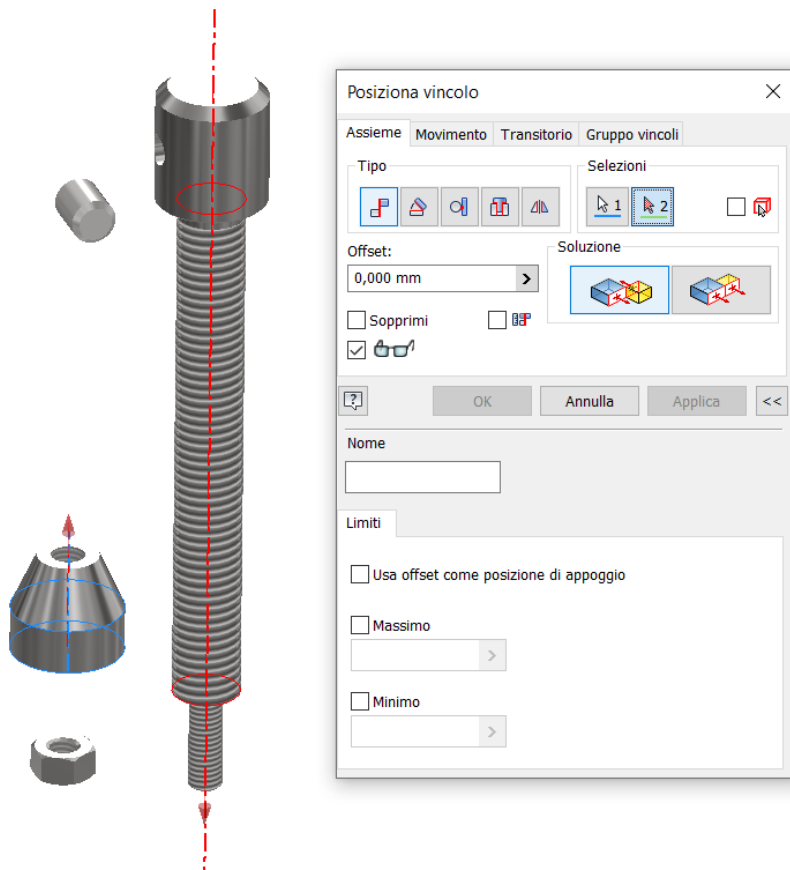## Sliding joint (the workpiece can only move in the groove)
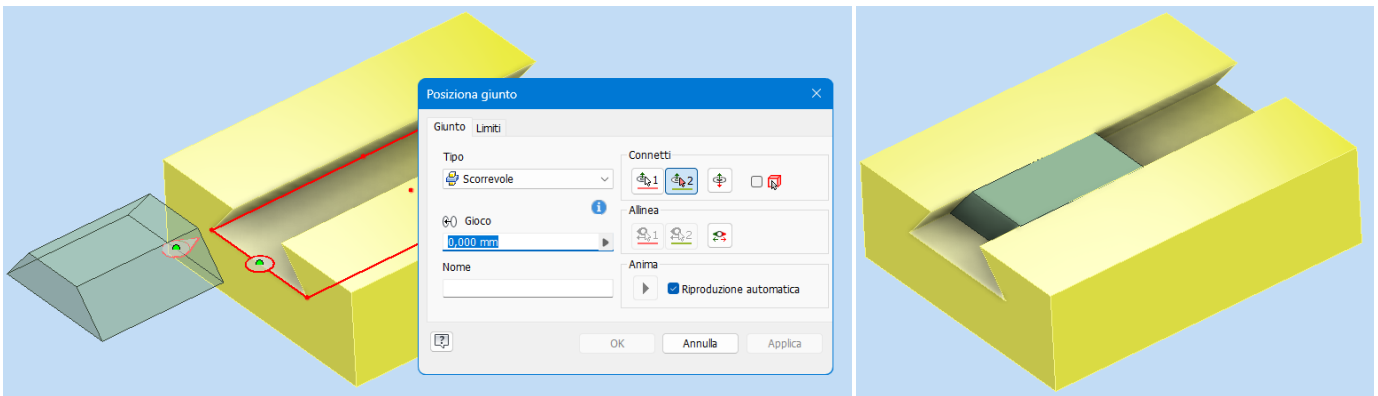
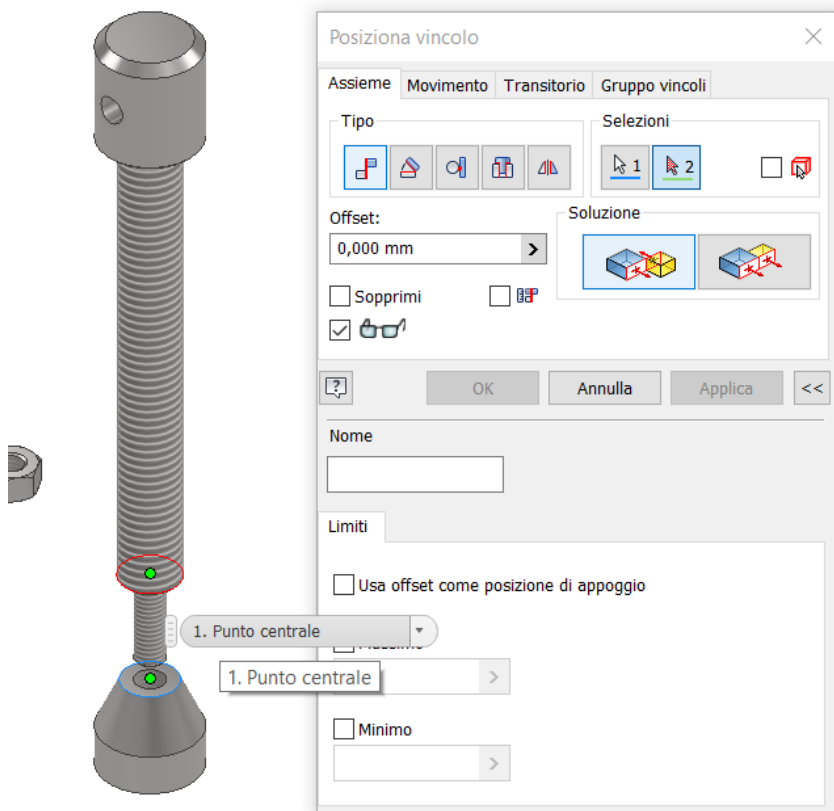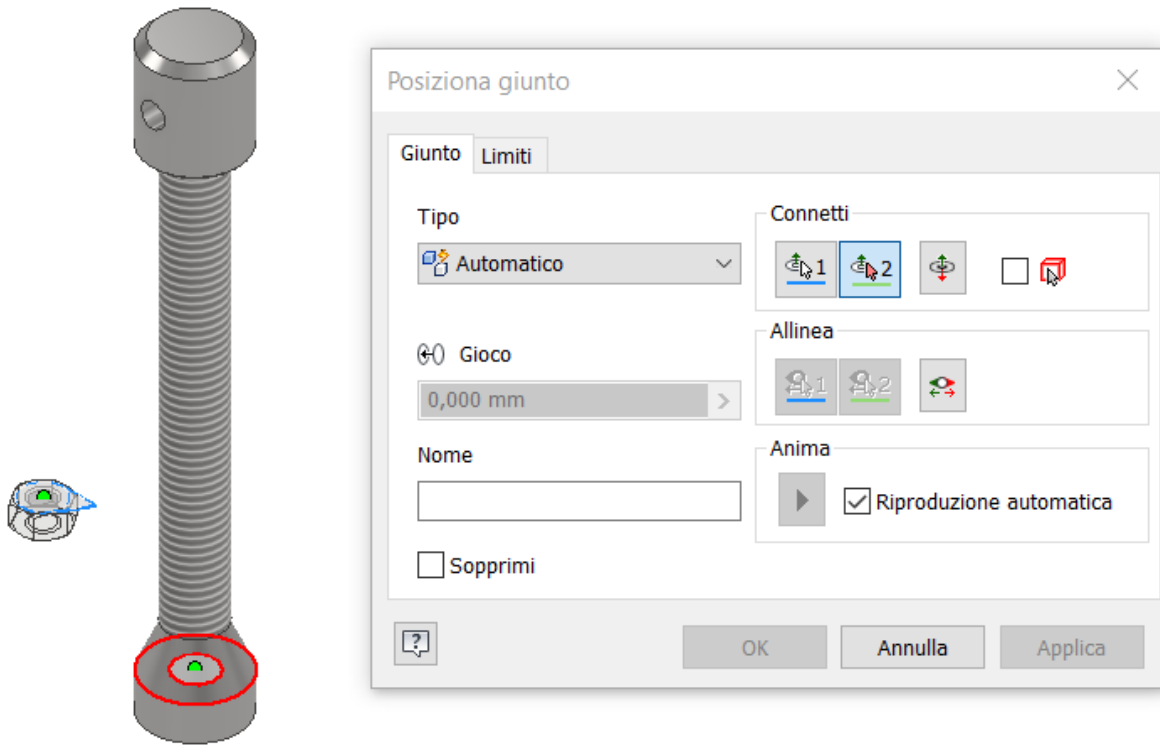## Rotational joint (screw can only rotate)
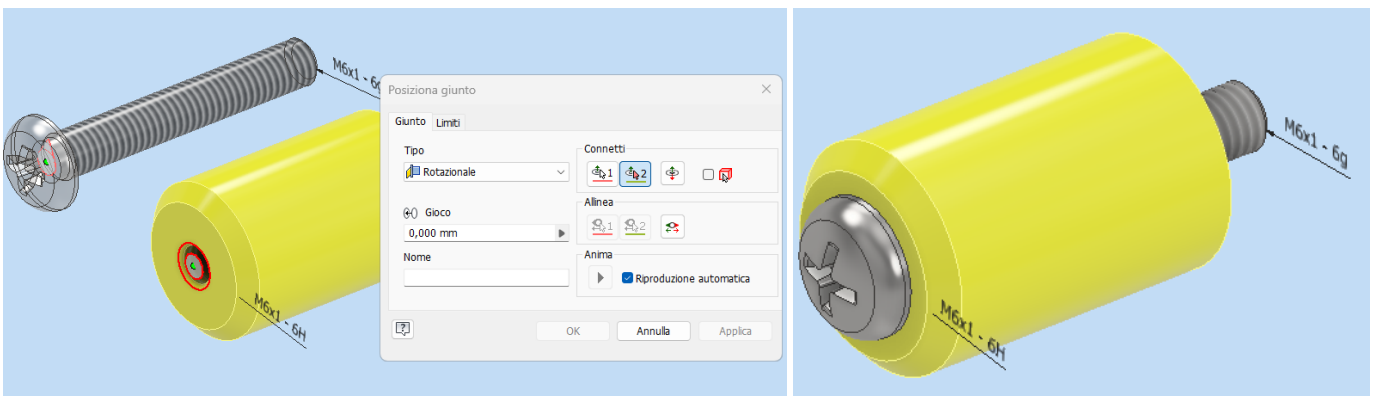
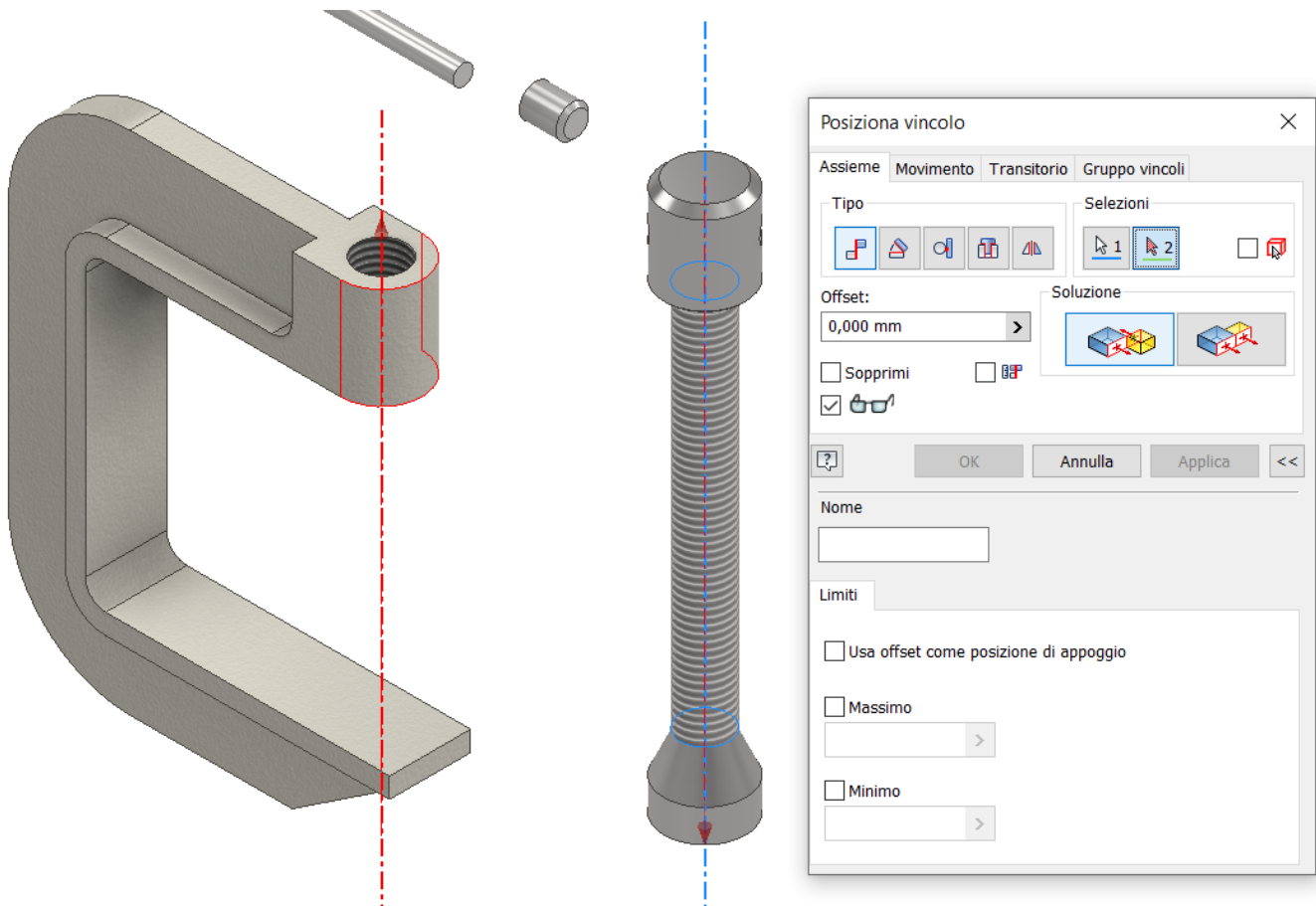It starts with an axial coincidence constraint between the axis of the clamp cap and the spindle axis

Circumference (or surface) coincidence constraint between clamping cap and spindle
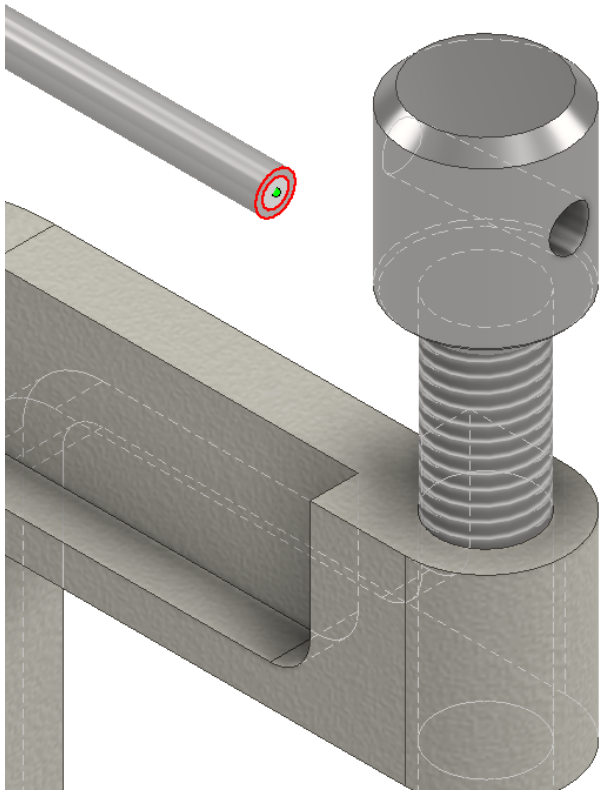
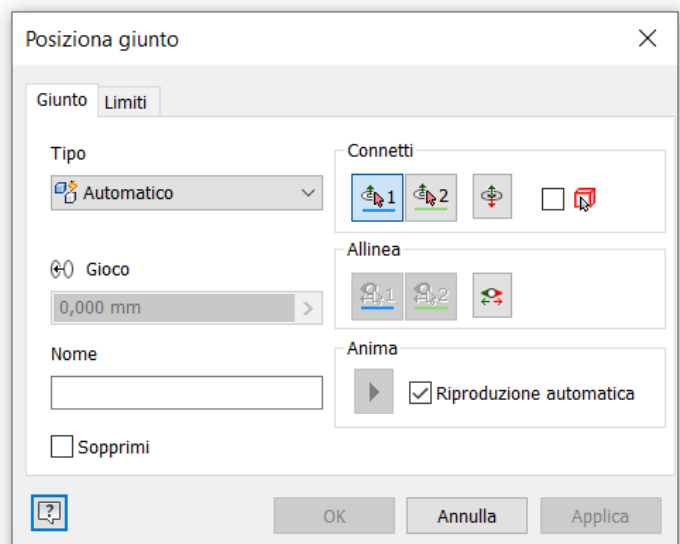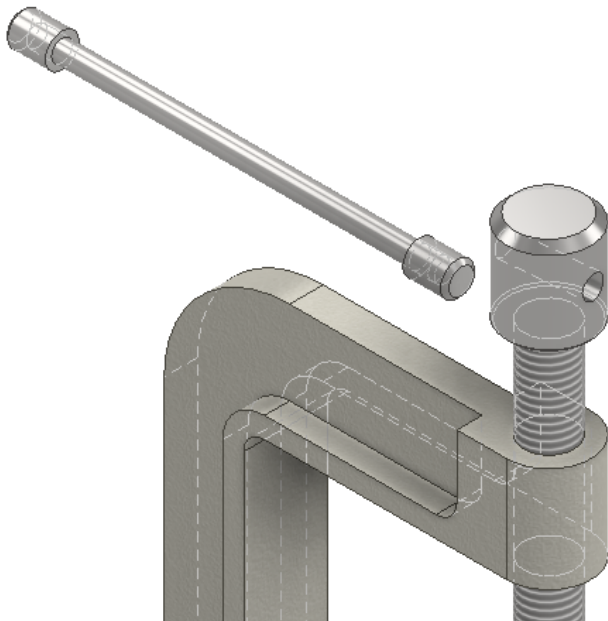Automatic joint between nut and spindle end screw (the joint is a set of simple constraints)

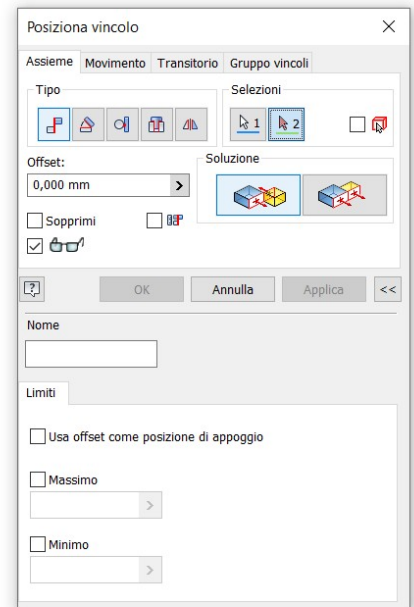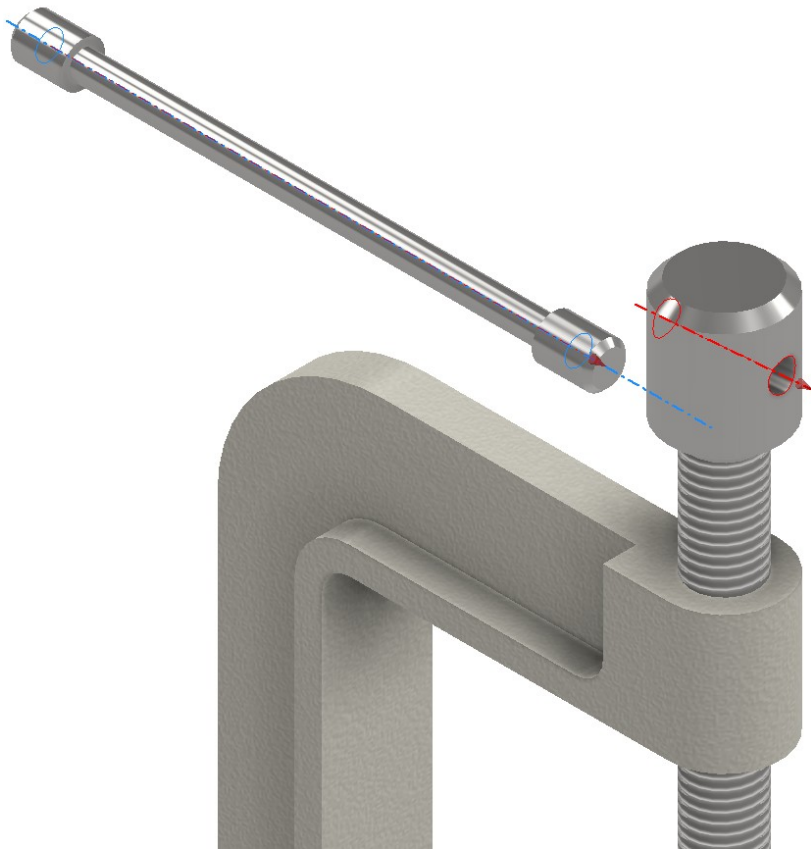Spindle axis constraint with vice bore axis

Automatic joint between handle cap and handle (or axial constraint and then surface or circumferential constraint).
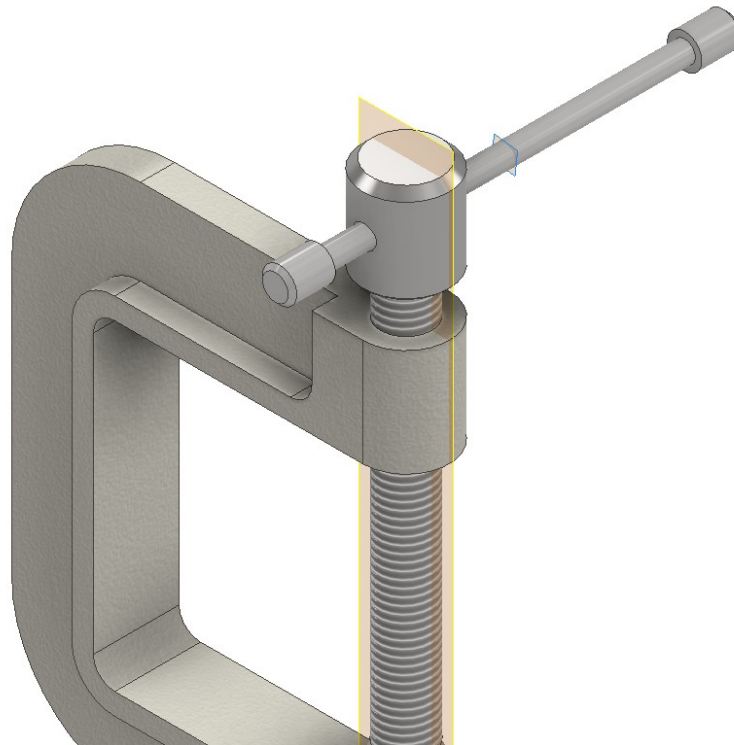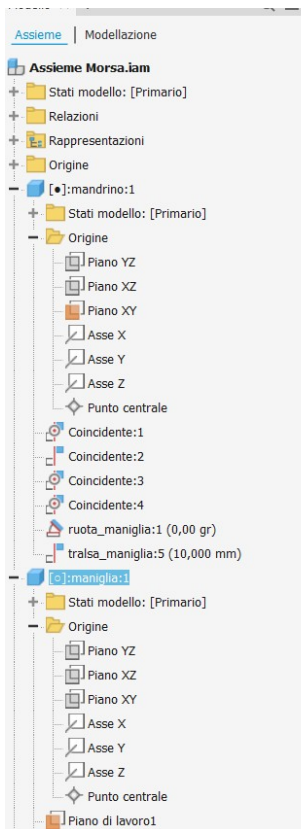 NB: the display of hidden edges has been enabled for ease of selection.
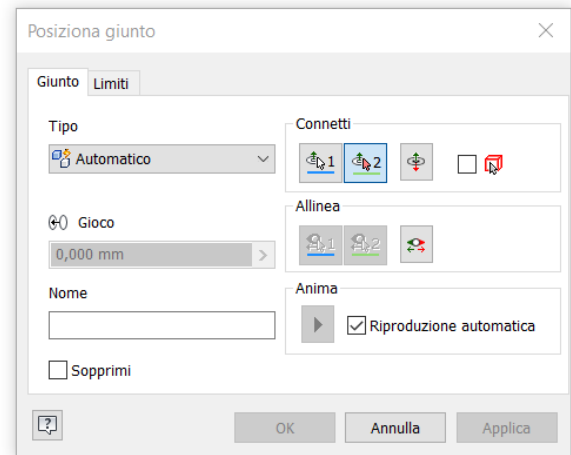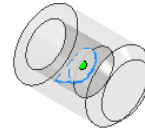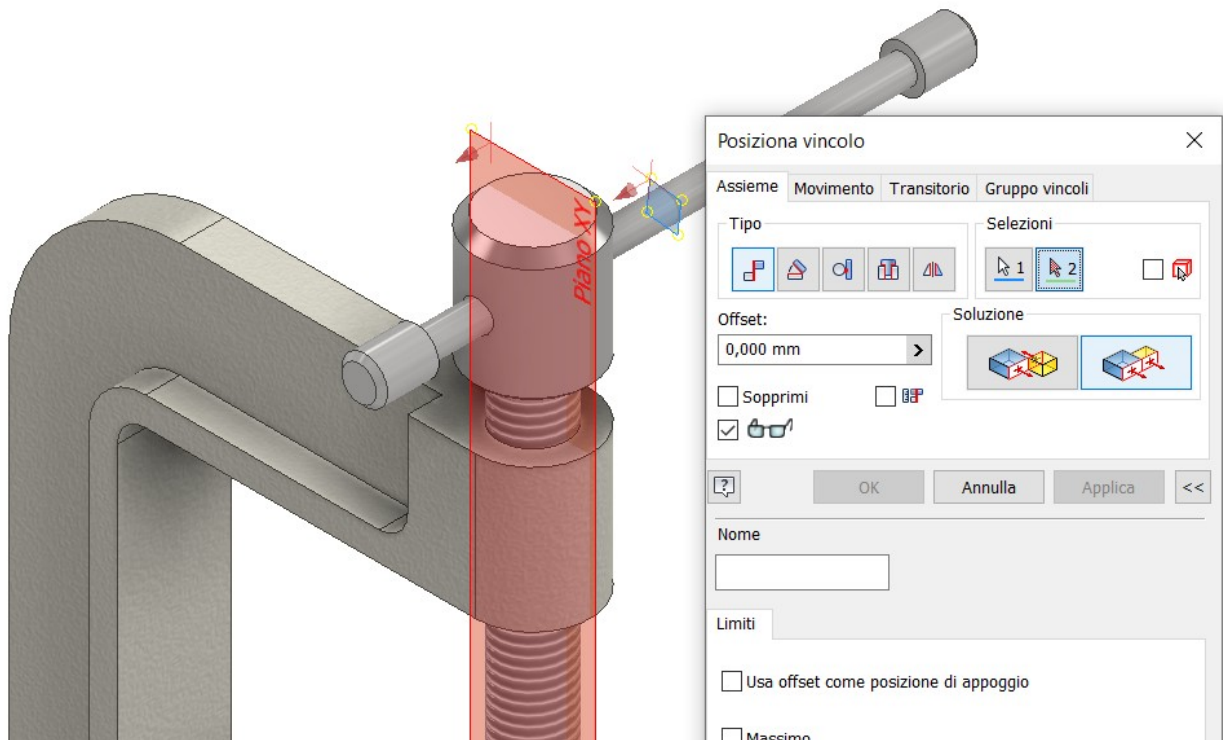

Repeat for the second cap

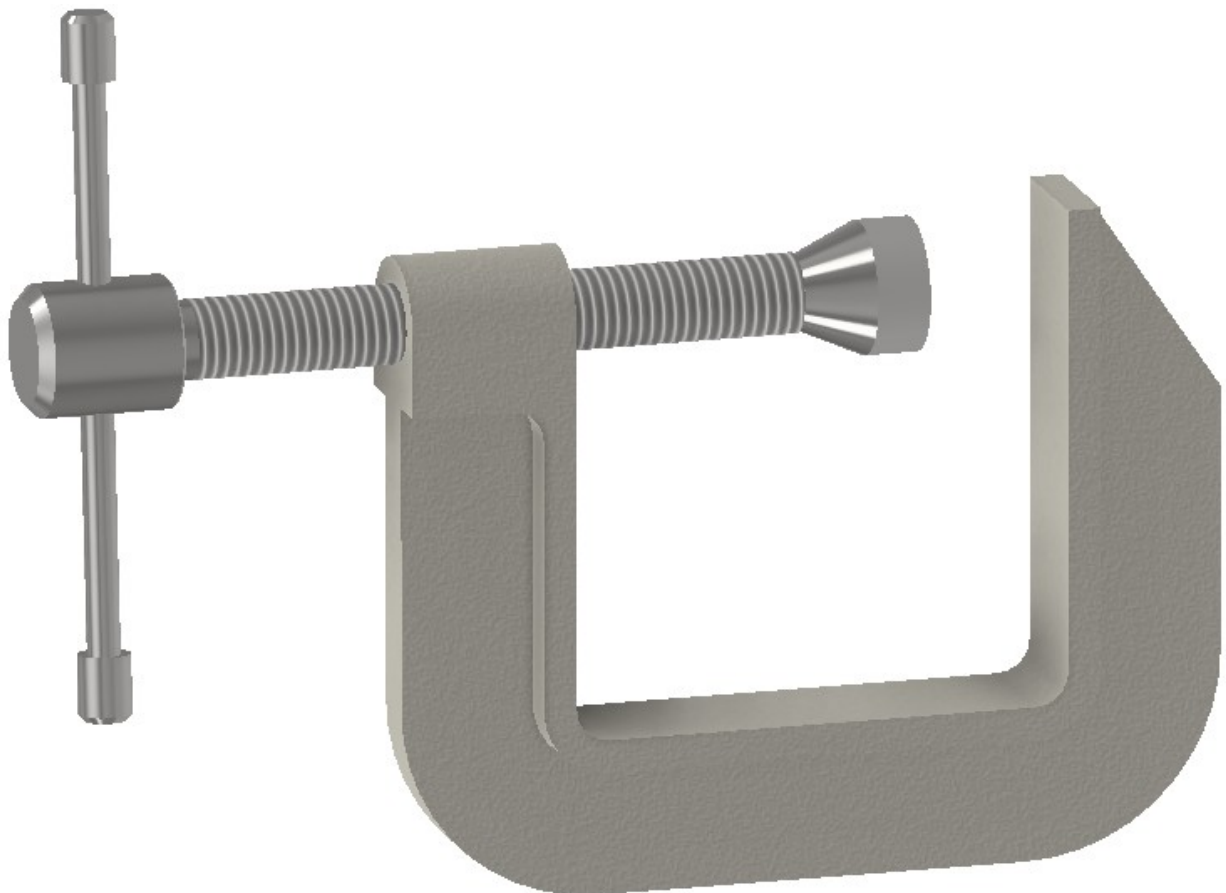Axial constraint between handle and spindle bore axis

Display symmetry planes of handle and spindle

Insert a coincidence constraint between the two planes of symmetry to lock the handle in the spindle hole

Final assembly

ANIMATION OF ASSEMBLY PARTS

We want to rotate the spindle in the threaded hole of the vice.
Animations are based on the rotation of a plane of symmetry of the rotating part with respect to the fixed part.

We visualise the XZ plane of the handle in the parts tree and impose a parallelism constraint with the horizontal plane of the vise face.
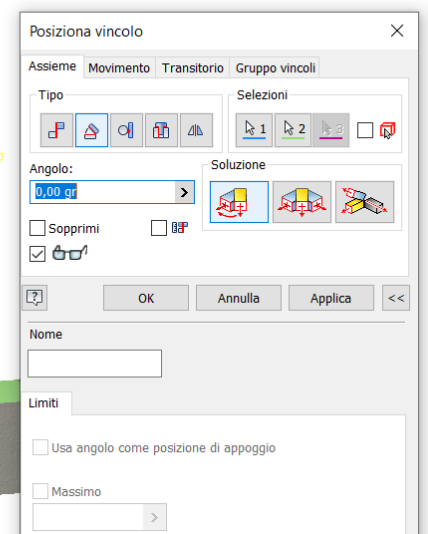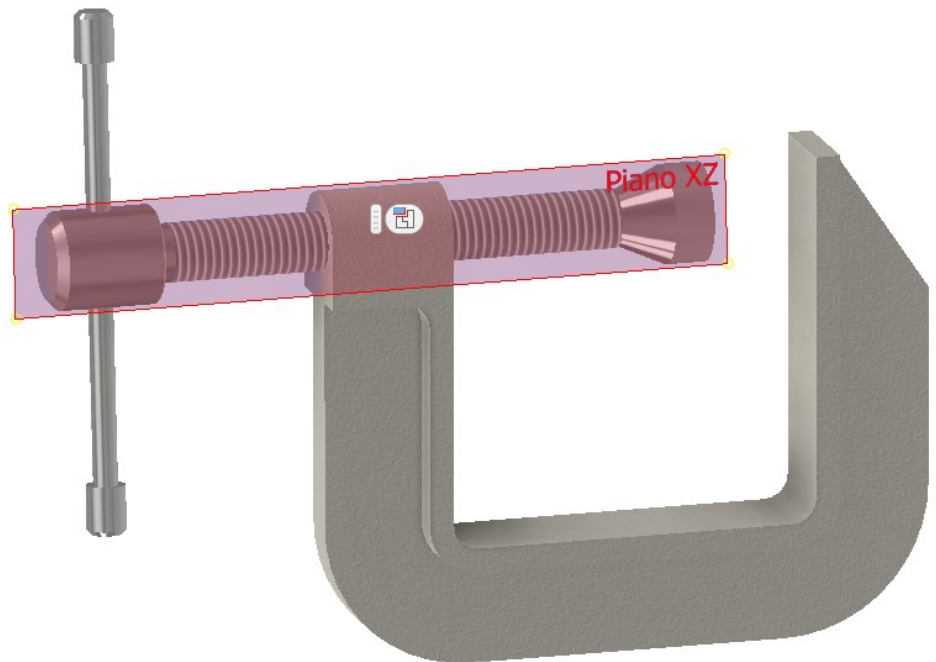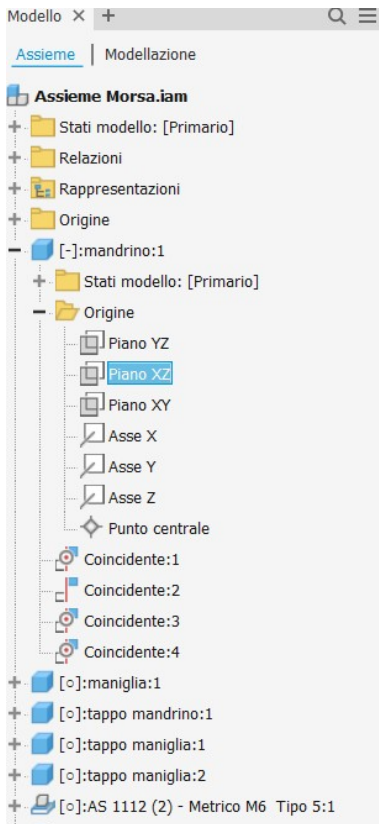
Rename the created constraint name so that it can be quickly identified in the tree (wheel_handle).
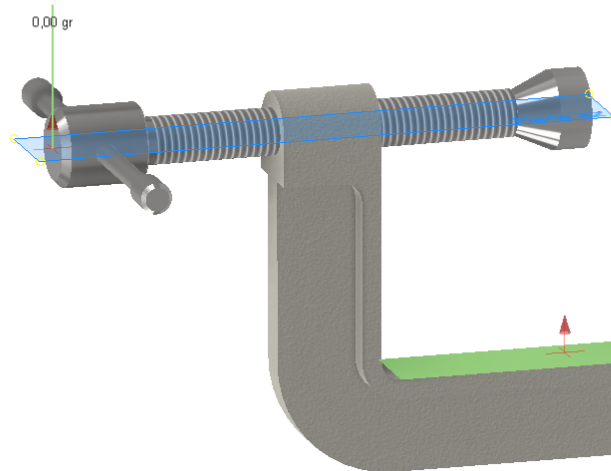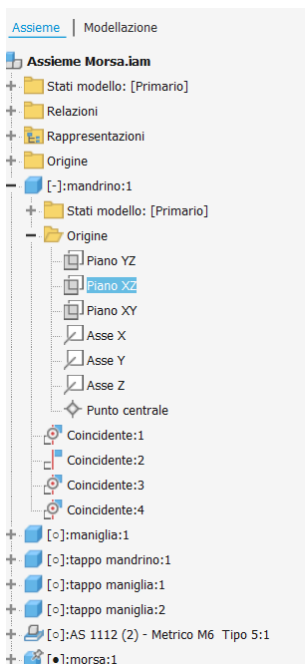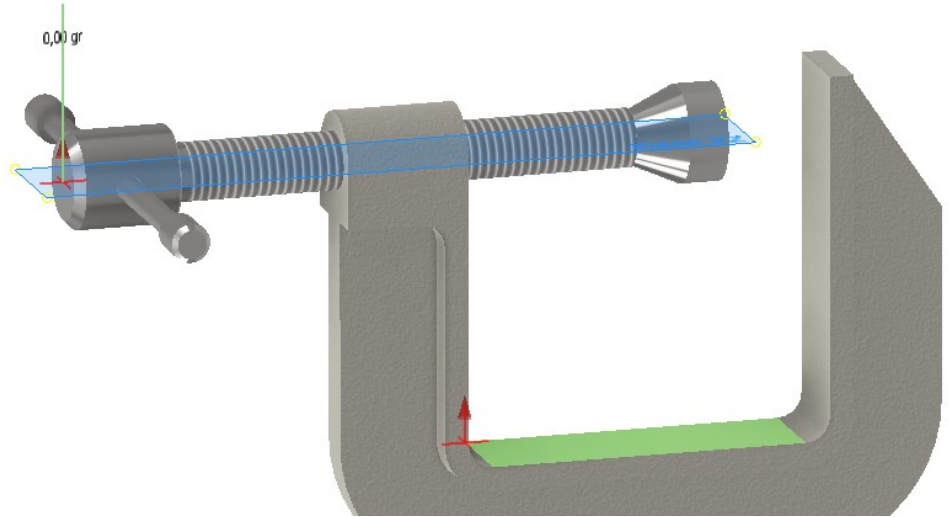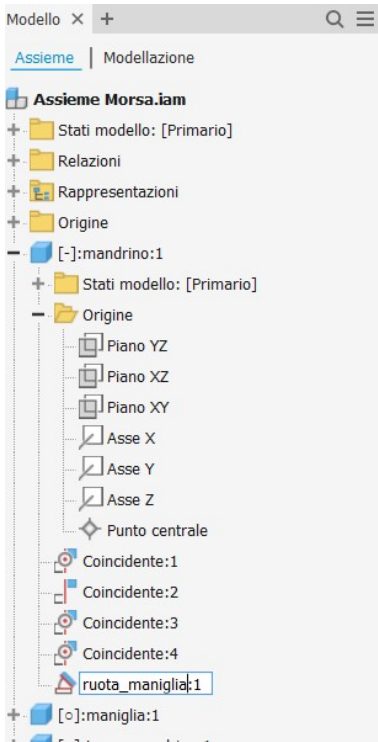

You can now 'animate' the constraint and display the spindle rotation
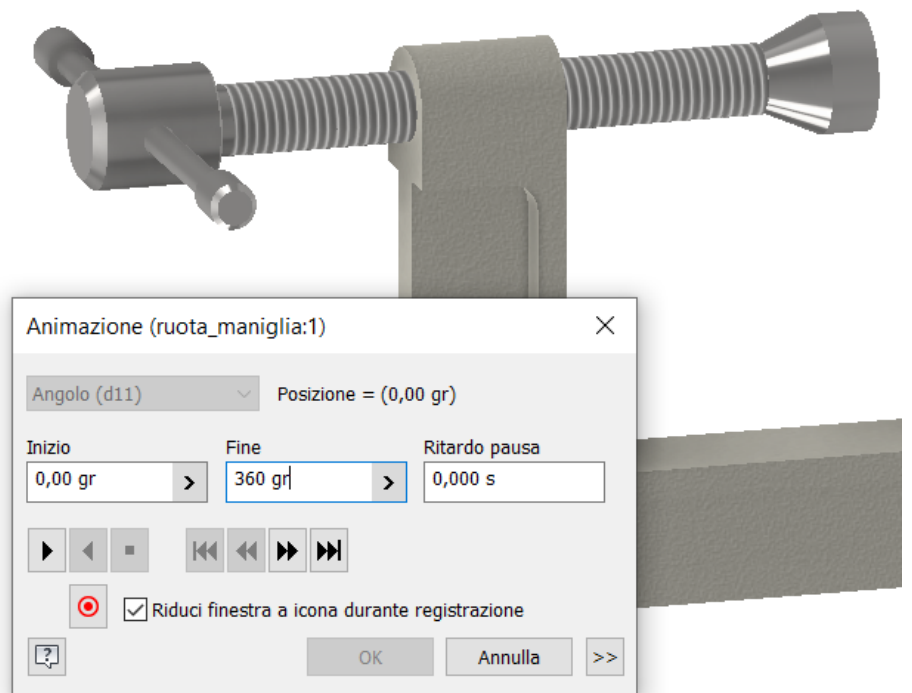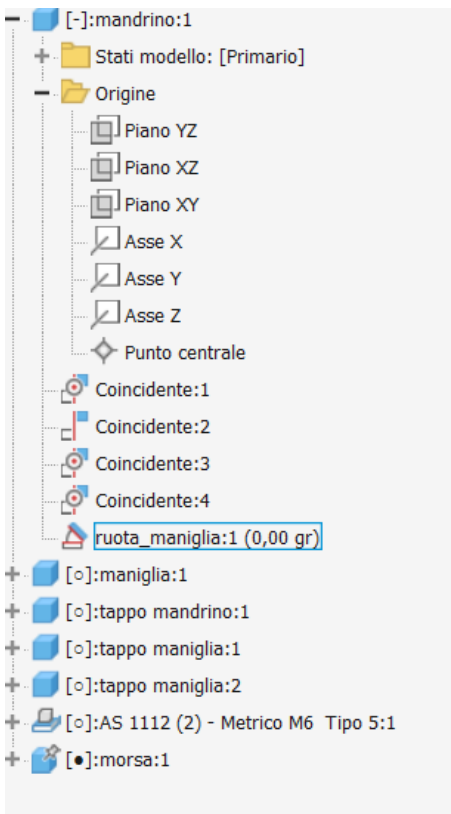
The spindle will rotate in the hole but will not translate. It is necessary to insert another constraint for translation.

Insert a coincidence constraint between the two faces highlighted in the figure.
The face in green can be selected by rotating the assembly or by pressing 'CRTL+A' to show the quick selection menu of available selection elements under the mouse pointer.
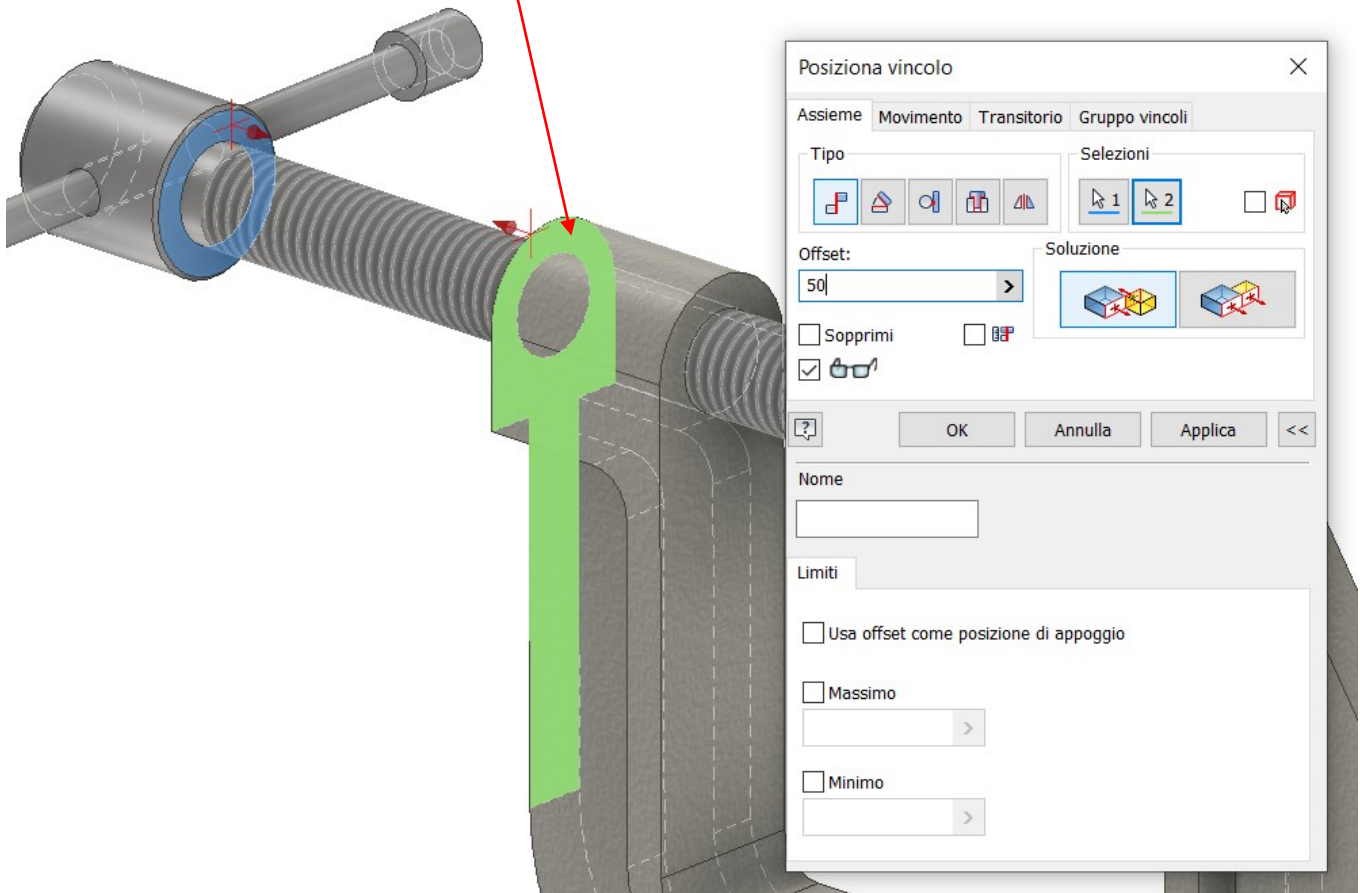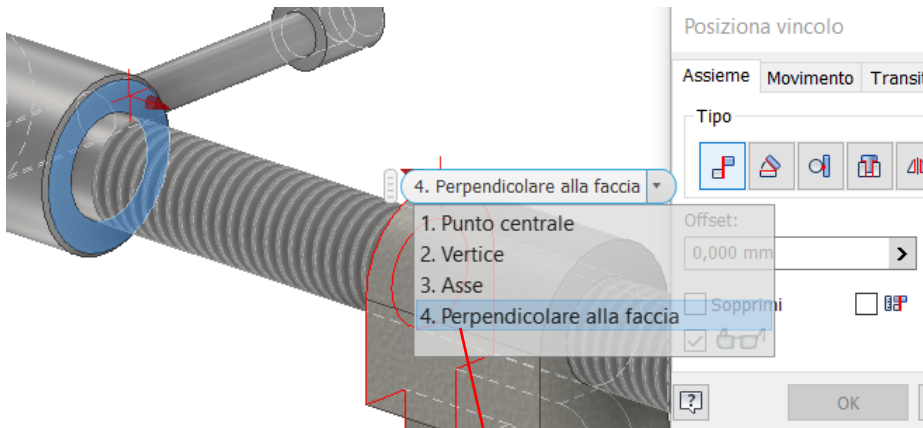
Rename the constraint for translation in the tree to 'translate_handle'.
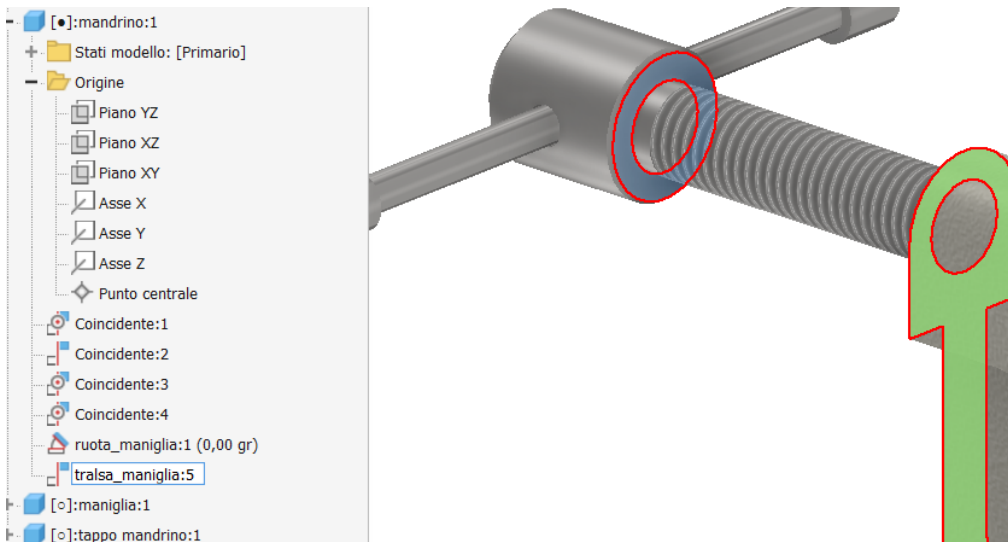
It is now possible to 'animate' the constraint and display the spindle translation.
The translation speed can be adjusted by expanding the animation mask.
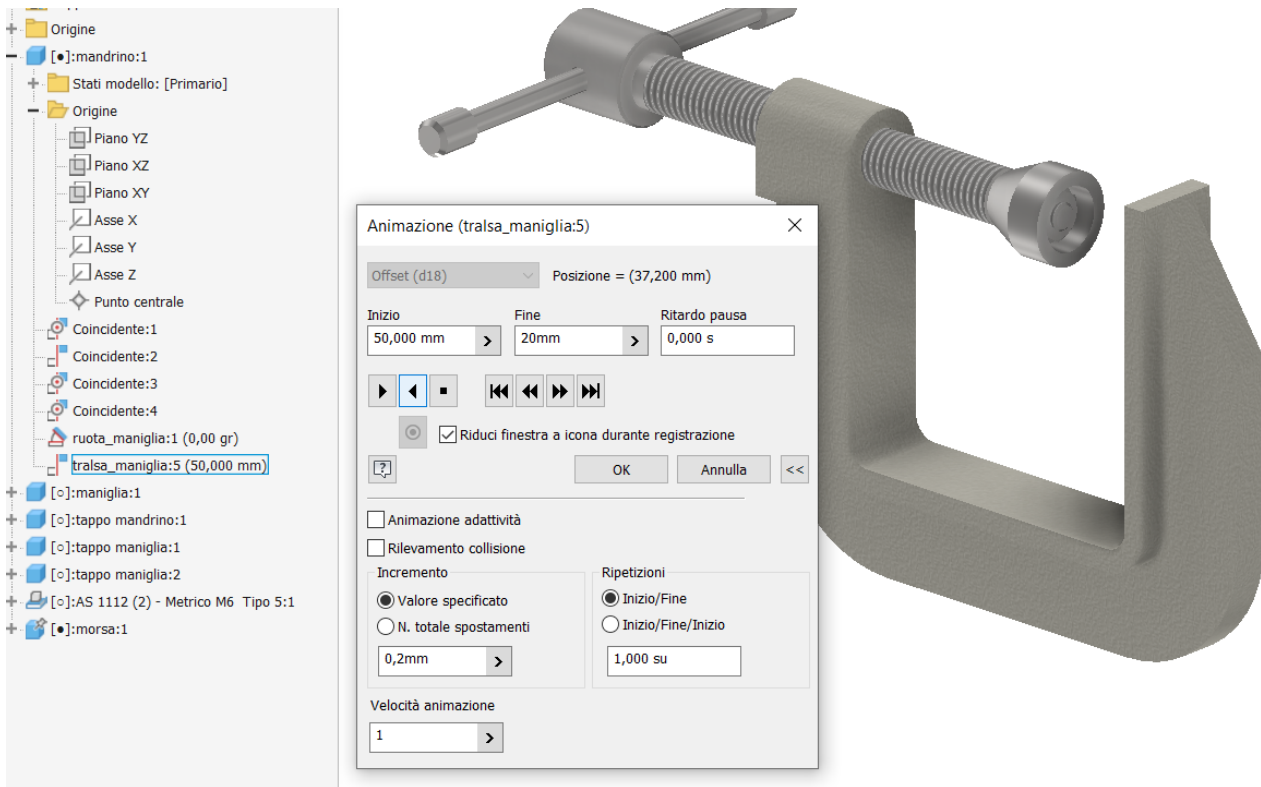
It is not possible to animate two movements in the assembly at the same time. For this purpose, you have to build the animation in the 'Inventor Studio' module)
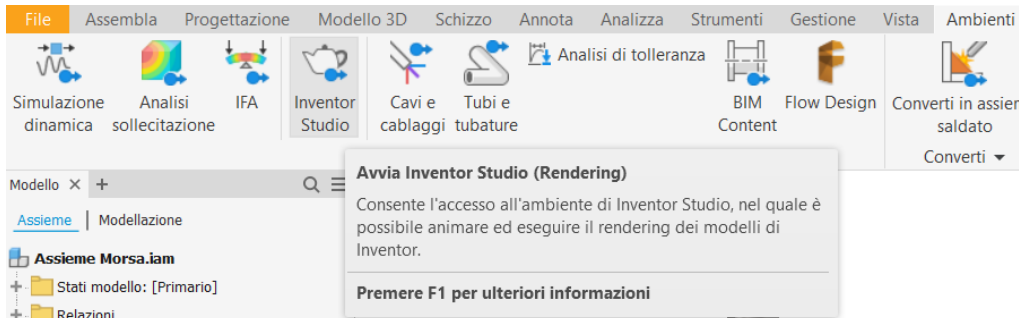
After opening the Inventor Studio module, create a 'linear animation representation'.
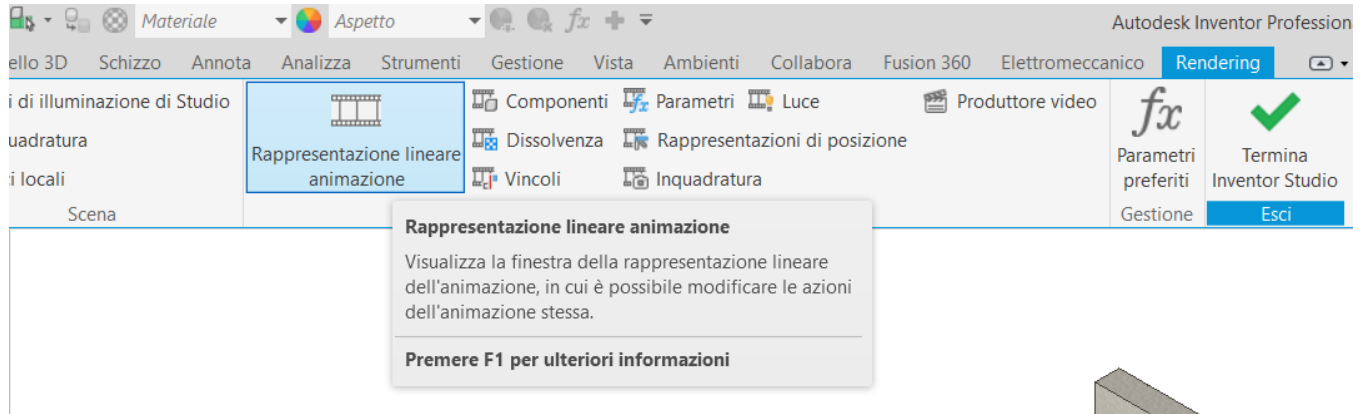
Set the approximate duration of the animation

Expand the 'linear animation representation' mask where we are going to add the individual animations.
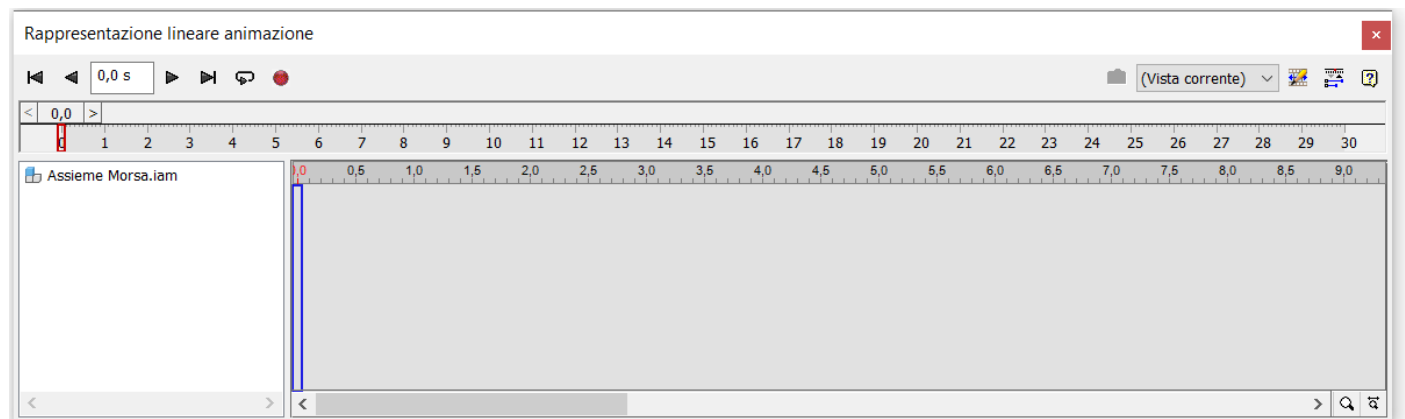
Locate in the assembly tree the part to be animated and select the 'wheel_handle' constraint to be animated.
Right-click to display the menu and click on "Animate constraints".
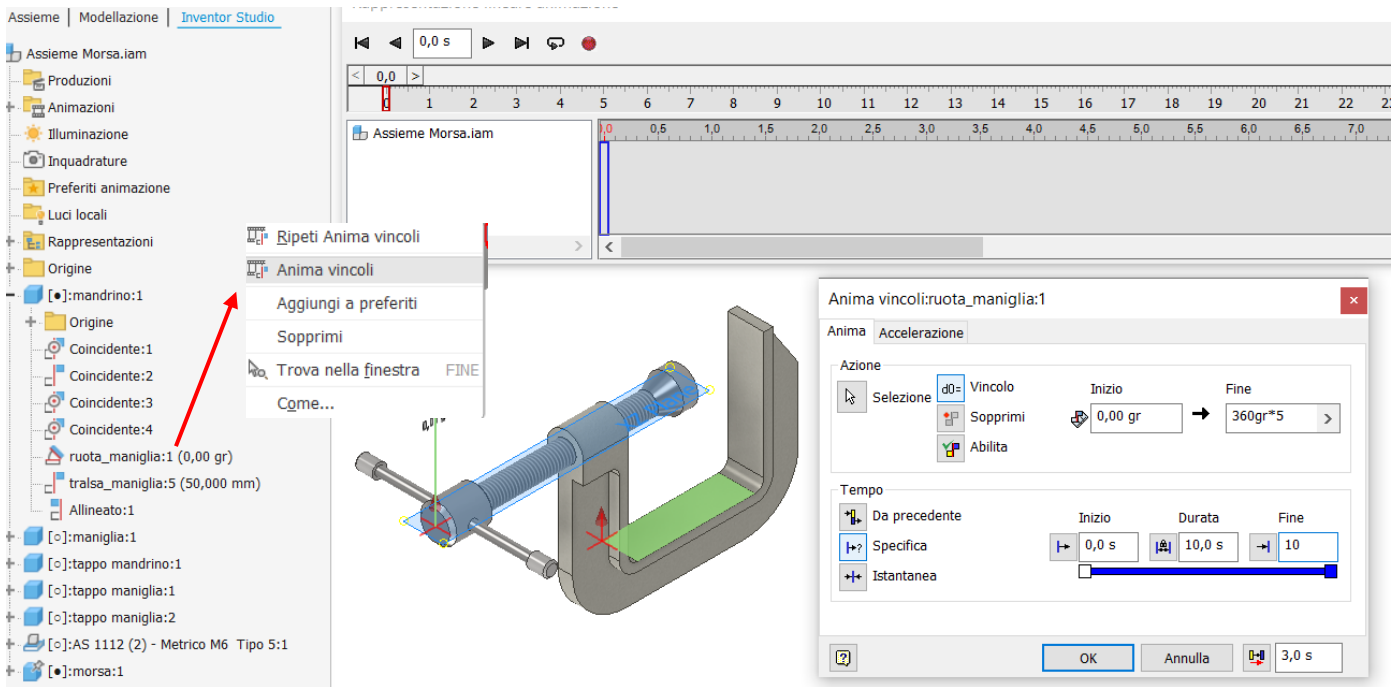
In the "Animation constraints" mask, set the constraint rotation (e.g. 360°*10) and indicate the start and end time (specific).
By clicking OK, a trace will be added to the "animation linear representation" mask, which can be executed by clicking on the play button.
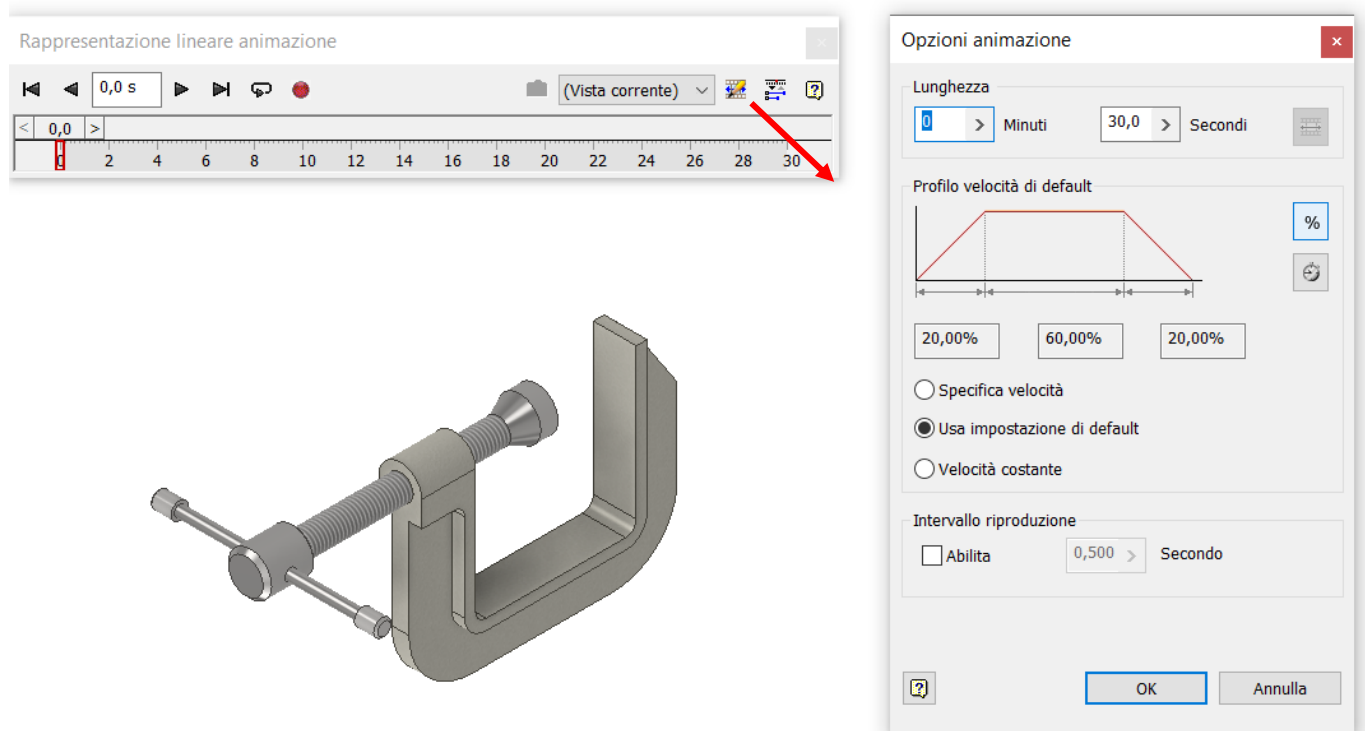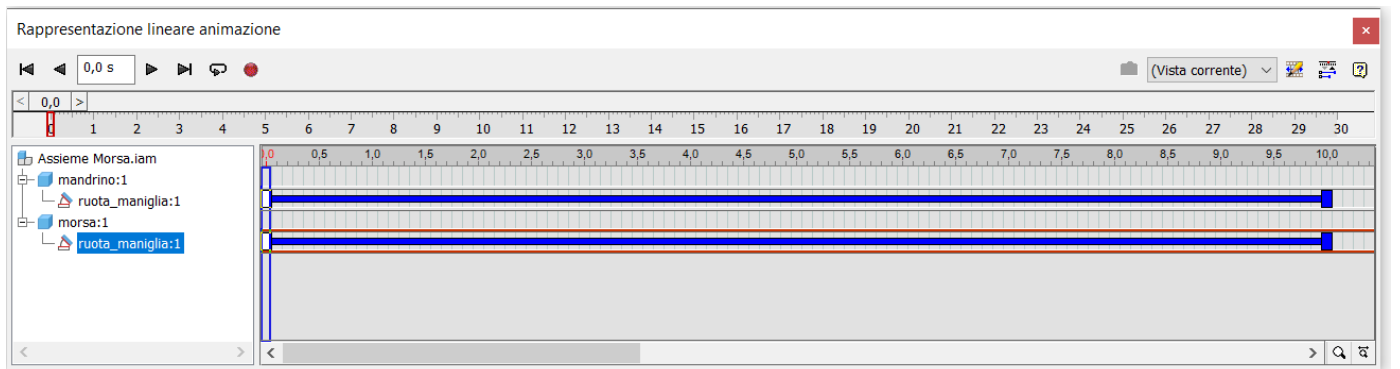

Proceed by selecting the second constraint 'move_handle' to be animated as done for the previous one.
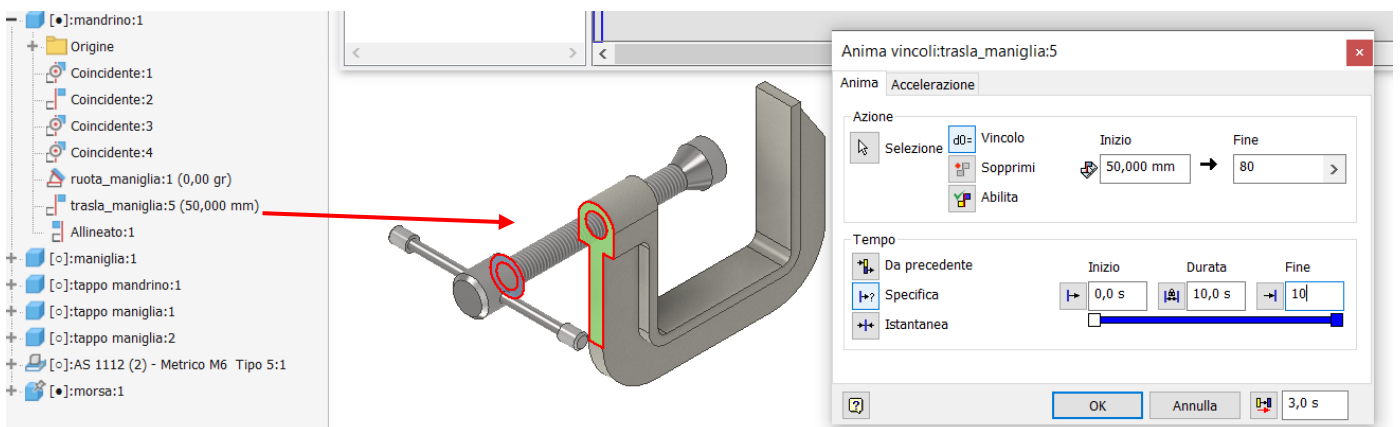
Continue construction by returning the spindle to its initial position.

Having set in the rotation constraint 10 revolutions (360gr*10) in 10 seconds then the spindle will have to translate by the pitch (1.75mm) for 10 revolutions i.e. 17.5mm.

To make the animation realistic, the formula must be set in the translation distance of the constraint:

50 mm - 10 mm * 1.75

By the end of the rotation, the spindle will have moved 17.5 mm.
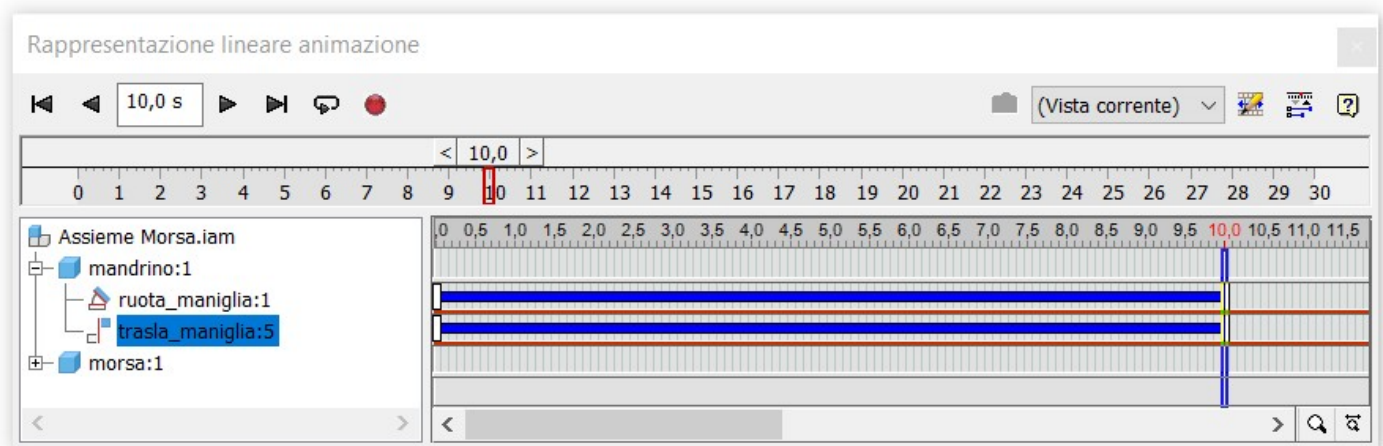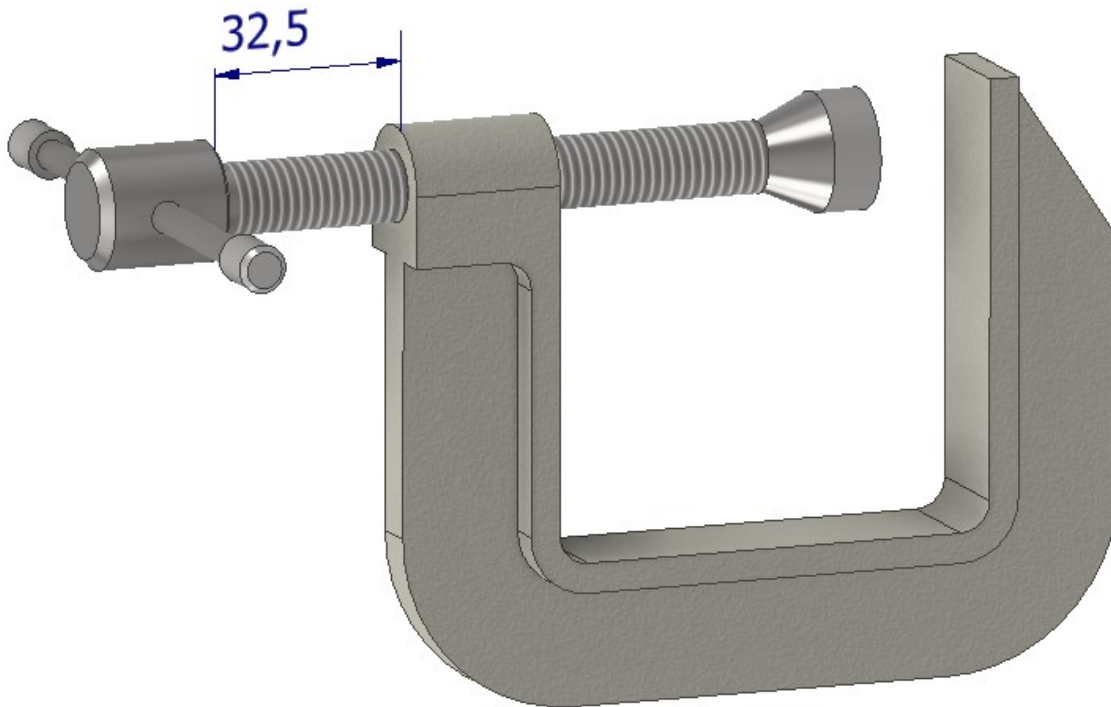
KINEMATICS VS DYNAMICS

Animations made with 'Inventor Studio' are kinematic, i.e. they do not take into account masses, inertia, friction, etc. .

A presentation file is used to create animations and exploded views of the structure.
It is very useful for showing the order of the parts to be assembled or the relationships between the various components.

From the MENU "*file*" "*new*" you reach the mask for creating a presentation.
Select "standard" and press "create". You will be asked to select an assembly file.
Immediately save the file with a unique name identifying the presentation
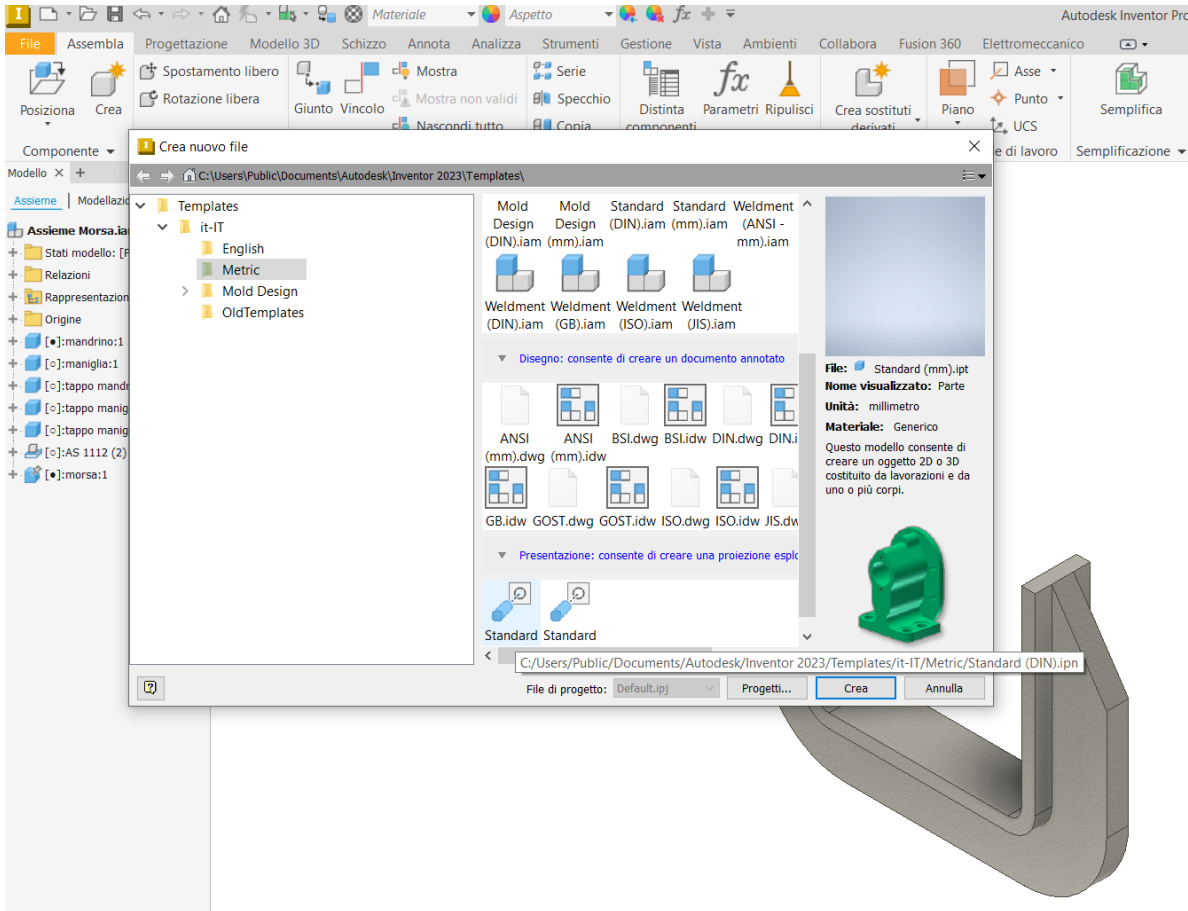
Right-click on 'Component Position Adjustment'.

Select the spindle to display the triplet of Cartesian axes that will be used to move the part.

Grasp the horizontal arrow with the mouse (it becomes coloured) and move the spindle to the left.
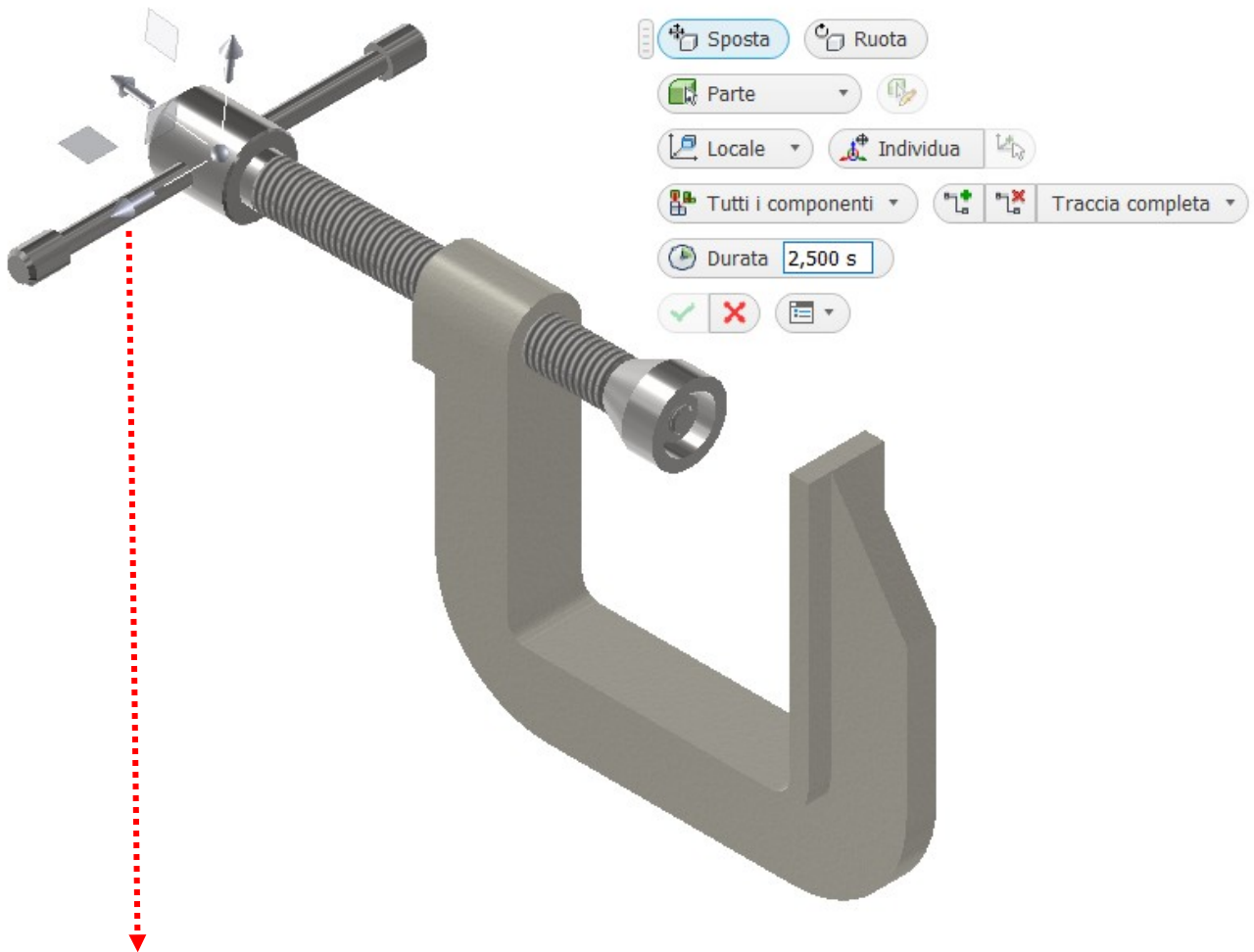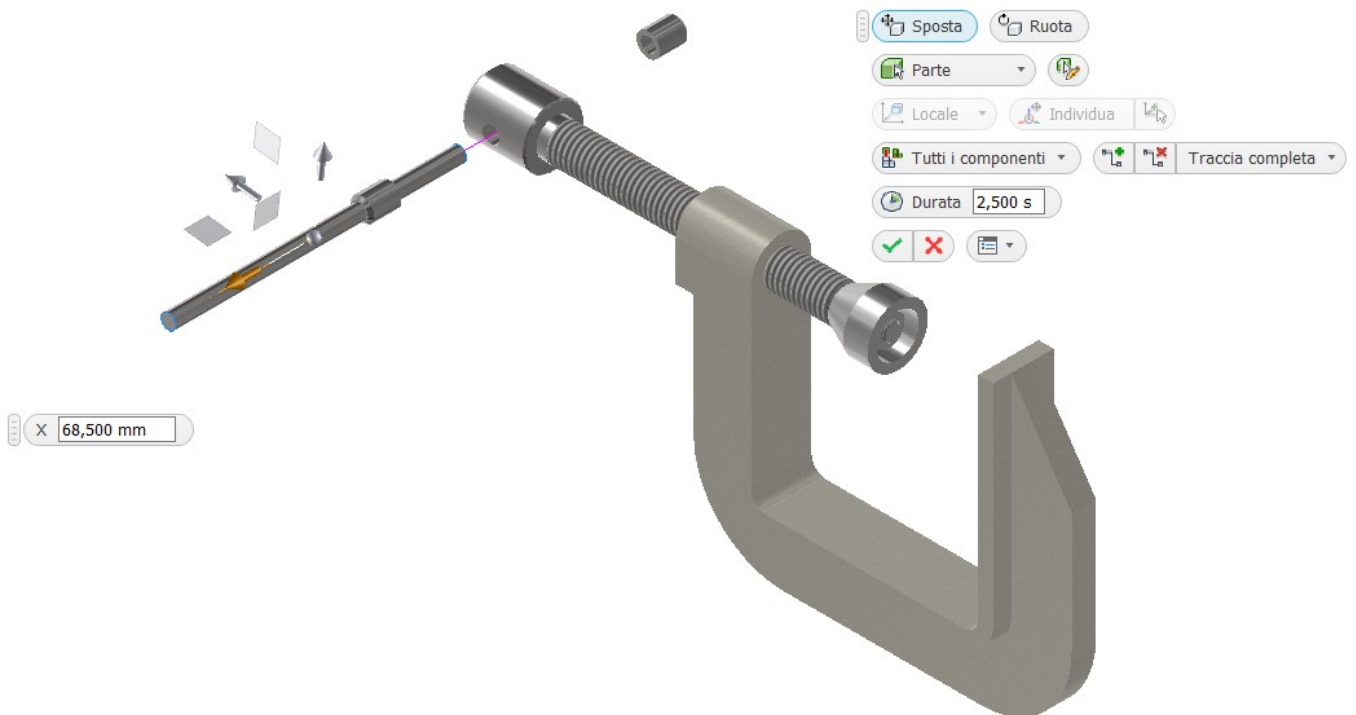
At the bottom of the window, this movement will become a 'Storyboard' sequence.
 At any time, it is possible to order, edit and animate the present actions.
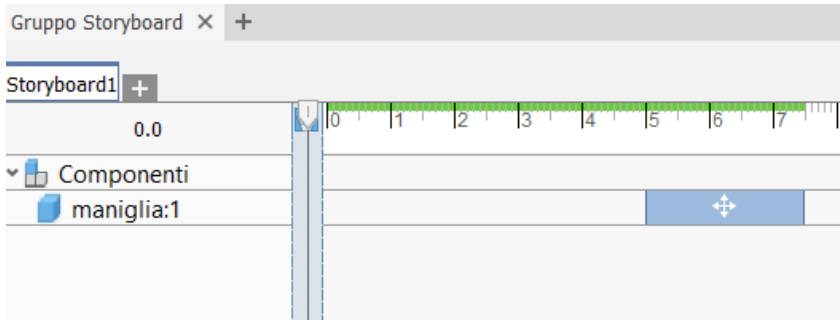
Continue by moving the various parts as in the figure in the order indicated in the Storyboard.

A video presentation of the assembly can now be created.

INVENTOR : EXPORT ASSEMBLY TO AN INTERCHANGE FORMAT

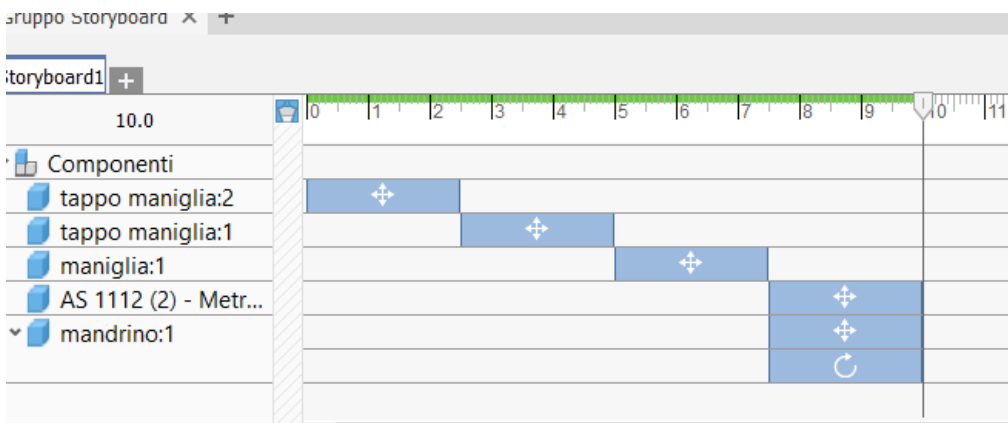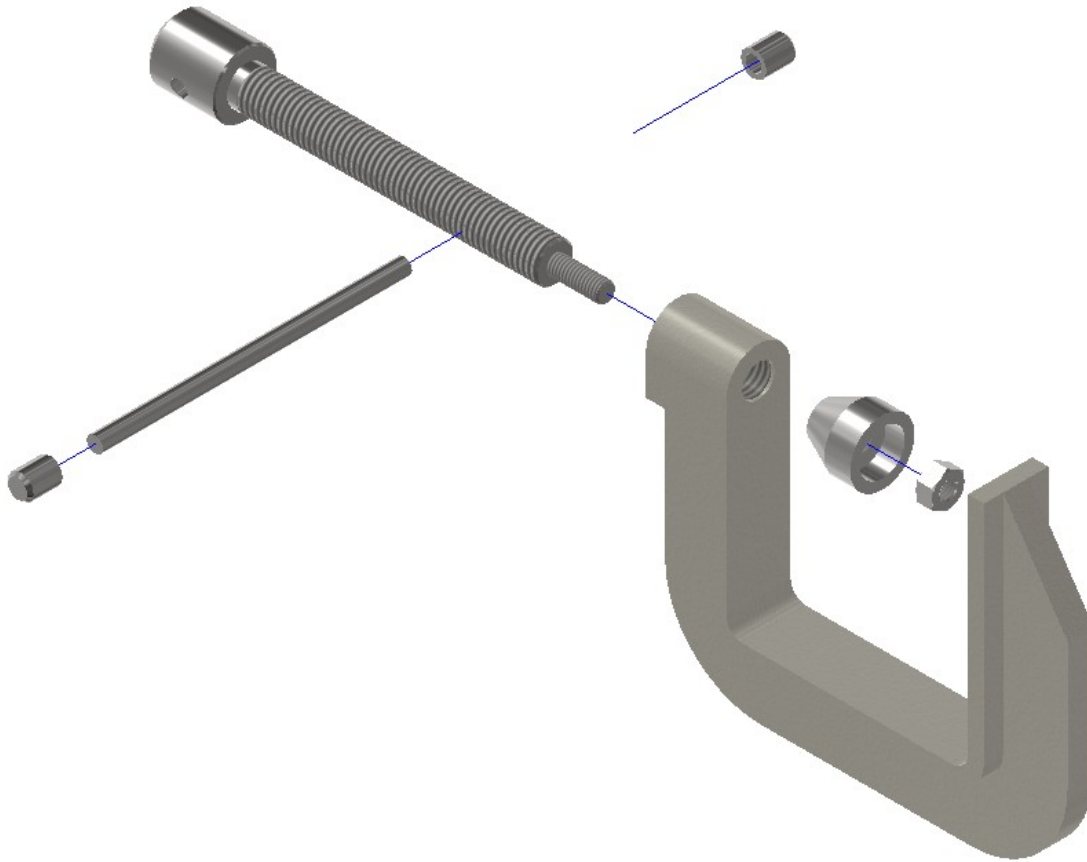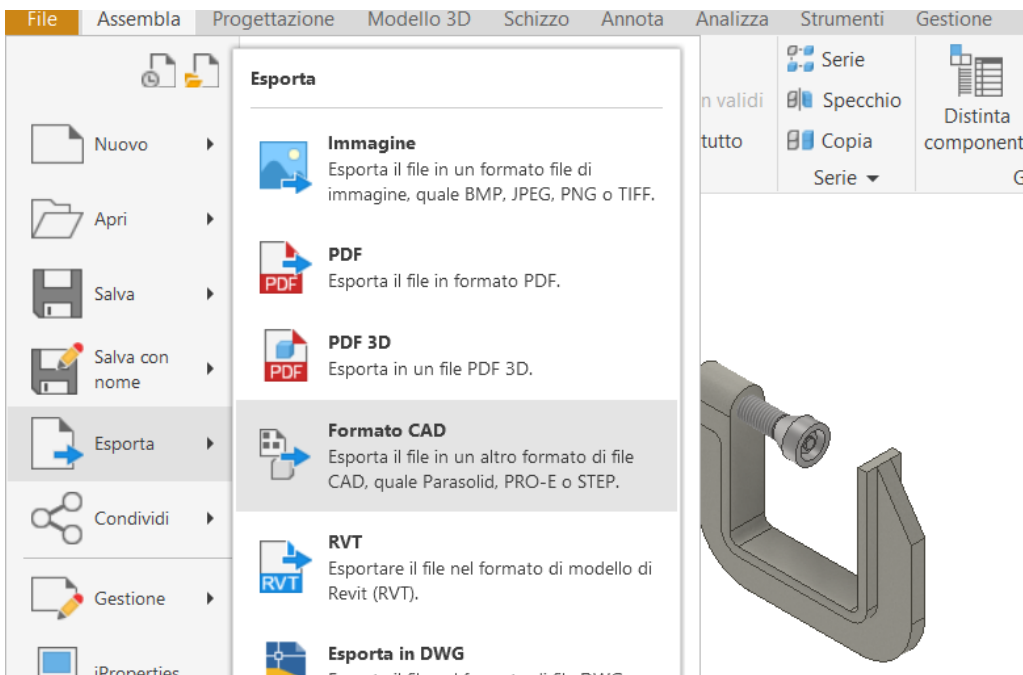When you want to share a 3D assembly made with Inventor with other users, you can compress the folder containing the project and provide the resulting compressed archive via e-mail, USB stick, etc. .

The user will only be able to open the drawing if they have the same or a higher version of Inventor than the one that generated it.

If the user does not have Inventor of the same or higher version (or if you do not want to share the assembly with all native parts in Inventor) then you must export the assembly in an interchange format:

- STEP (STandard for the Exchange of Product model data), is an ISO standard containing a set of rules for the integration, presentation and exchange of data (via computer); it can be used to transfer data between the following systems: CAD, CAM, CAE, PDM/EDM.

- IGES (Initial Graphics Exchange Specification), first published in January 1980 by the U.S. National Bureau of Standards, (now National Institute of Standards and Technology) defines a neutral data format that allows the exchange of data, graphics files and information between CAD systems.

Export in *step* format