WS 09.3: Wire Bonder

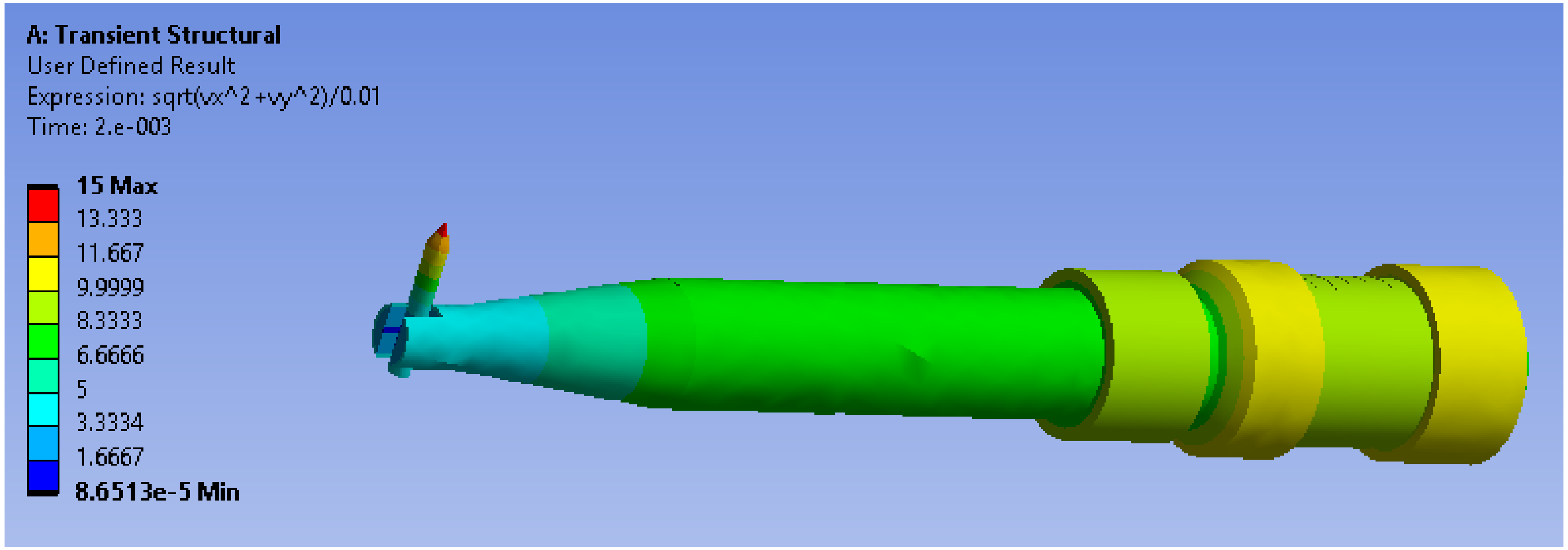
Release 2022 R2

Please note:
- These training materials were developed and tested in Ansys Release 2022 R2. Although they are expected to behave similarly in later releases, this has not been tested and is not guaranteed.
- The screen images included with these training materials may vary from the visual appearance of a local software session.
- Although some workshop files may open successfully in previous releases, backward compatibility is somewhat unlikely and is not guaranteed.

Ansys

Workshop 09.3 - Goals

- The objective of this workshop is to illustrate the process of applying initial angular velocity to a wire bonder using Steps.
  - Currently Mechanical doesn't offer Angular Velocity as an Initial Condition other than within an Explicit Dynamic analysis.
  - Initial angular velocity = 15 rad/sec. measured at the tip of the tool.

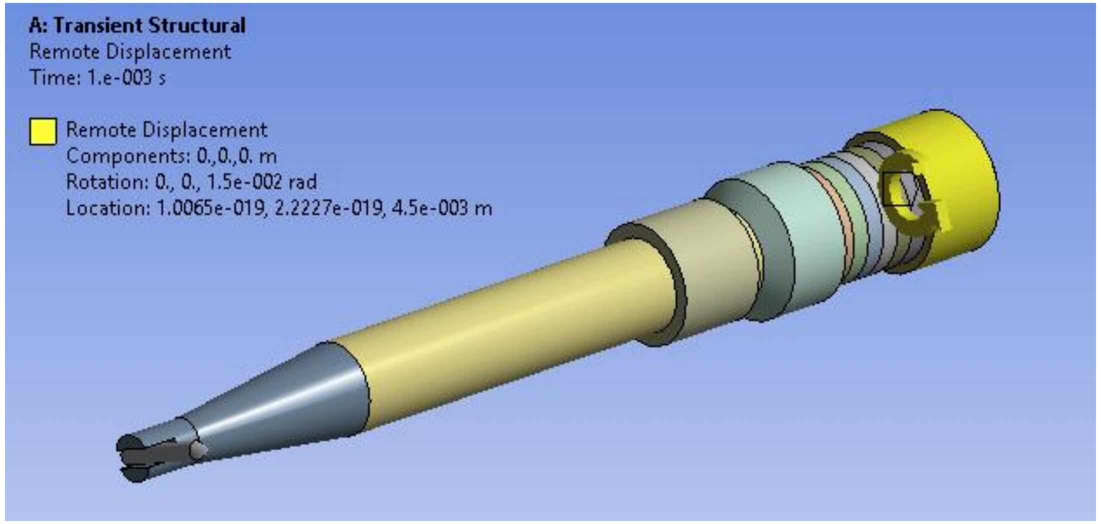


Workshop 09.3 - Project Schematic

- Start a new Workbench session and open the archive file “WS09.3-Wire_Bonder.wbpz” supplied with this workshop.
- Open the model in Mechanical.
- This workshop will be solved in Metric units:
  - Metric (m, kg, N, s, V, A)
  - Radians
  - Rad/s

Workshop 09.3 - Approach

- Impart the angular velocity by applying an angular rotation of 0.015 radians over a time increment of 0.001 seconds.
  - A two-step solution is required, with the first step conducted with Time Integration effects turned off.
  - Initial angular velocity of (0.015 rad/0.001 sec) = 15.0 rad/sec.
  - Use a remote displacement to apply the rotation, as it provides a means to access rotational degrees of freedom that otherwise aren't available within a solid model.
  - Apply it to the outer diameter of the tool near the global origin.



Workshop 09.3 – Analysis Settings

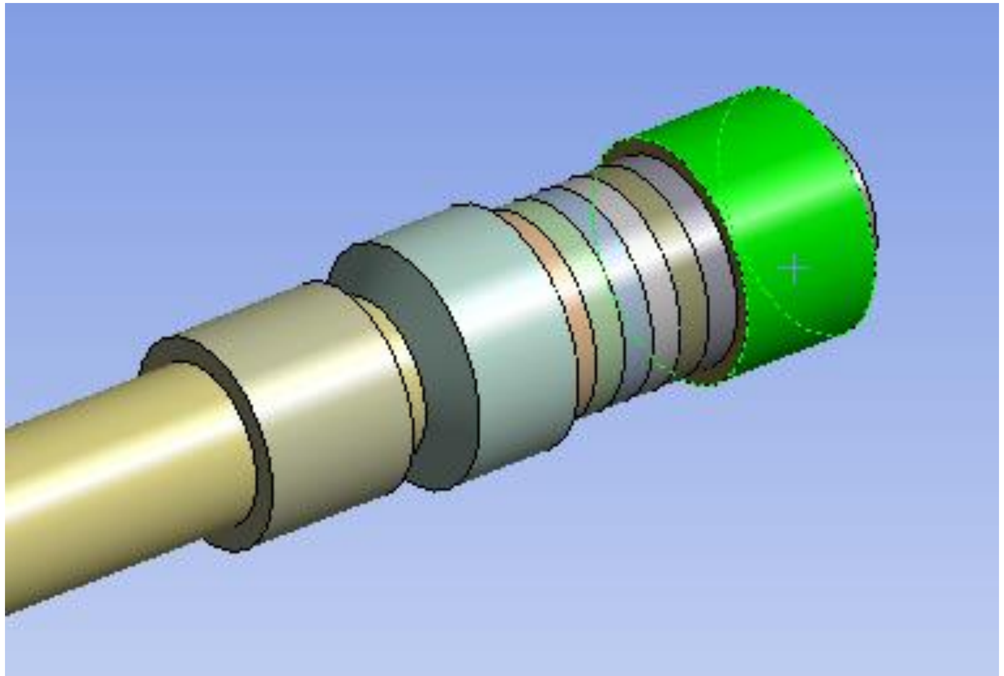
- Define a two-step solution, with the following step controls for each step:
  - Step 1
    - End Time = 0.001 seconds
    - Automatic Time Stepping = off
    - Substeps = 4 (1 substep would work, however 4 provides more accuracy)
    - Time Integration = **off**
  - Step 2
    - End Time = 0.002 seconds
    - Automatic Time Stepping = on
    - Substeps = 50 Initial, 10 Minimum, 200 Maximum
    - Time Integration = **on**

Analysis Settings

Properties	Step 1	Step 2
Step Controls		
Step End Time	1.e-003	2.e-003
Auto Time Stepping	Off	On
Define By	Substeps	Substeps
Carry Over Time Step	N/A	Off
Number Of Substeps	4	N/A
Initial Substeps	N/A	50
Minimum Substeps	N/A	10
Maximum Substeps	N/A	200
Time Integration	Off	On

Workshop 09.3 - Loads

- Apply the remote displacement to the face shown:
  - Use the Tabular Data to define zero translational displacements, RotX = 0, RotY = 0, and RotZ = .015 radians
  - De-activate the Rotational Z displacement during step 2 so the part is free to rotate at the prescribed velocity!



Tabular Data

	Steps	Time [s]	X [m]	Y [m]	Z [m]	RX [rad]	RY [rad]	RZ [rad]
1	1	0.	0.	0.	0.	0.	0.	0.
2	1	1.e-003	0.	0.	0.	0.	0.	1.5e-002
3	2	2.e-003	= 0.	= 0.	= 0.	= 0.	= 0.	1.5e-002
*								

Workshop 09.3 - Loads

- In transient analysis, loads are step-applied by default. When the load is ramped instead for the first time, it is interpolated from zero to the value of the current load step, and not from the initial value of the degree of freedom from the previous load step.
- Insert a Commands object in the Transient environment to instruct the solver to ramp the loads instead of stepping them.
  - In the commands window, enter the command `kbc,0`

Commands

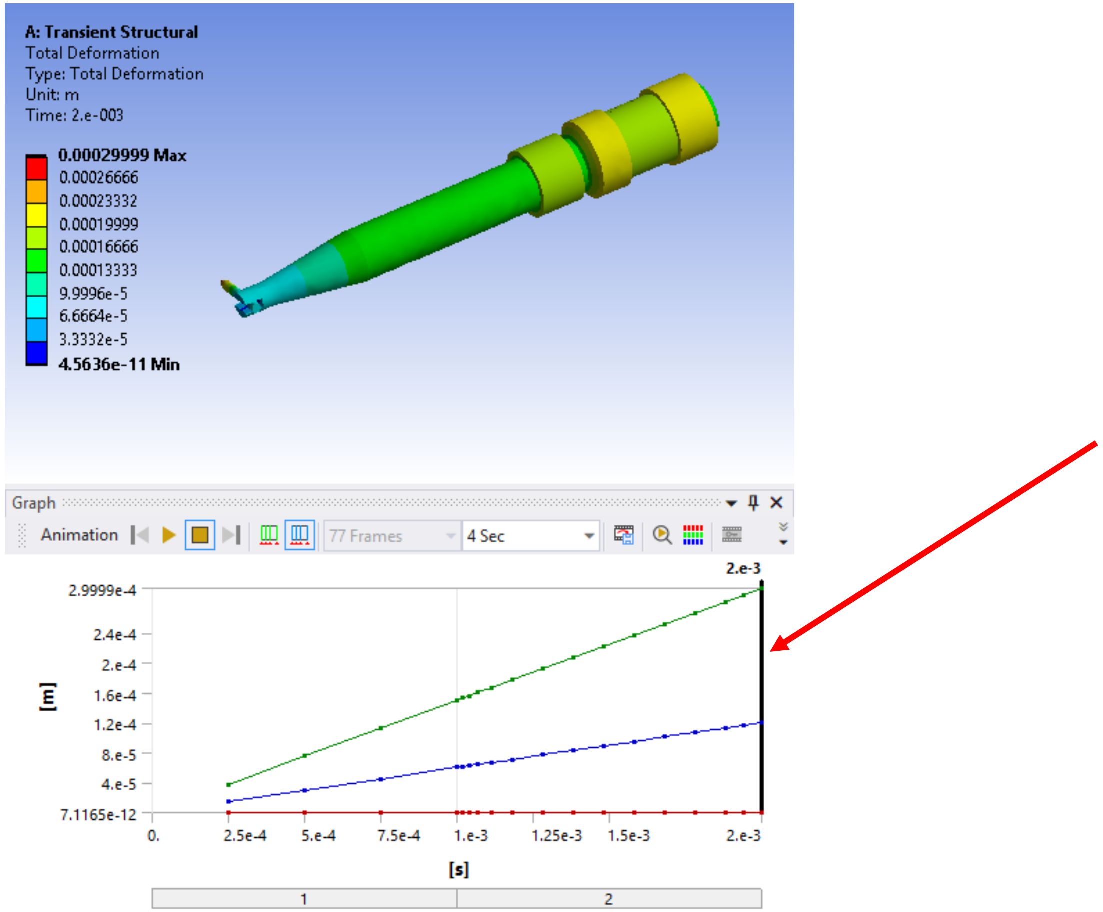
```
!   Commands inserted into this file will be executed just prior to the ANSYS SOLVE command.
!   These commands may supersede command settings set by Workbench.

!   Active UNIT system in Workbench when this object was created:  Metric (m, kg, N, s, V, A)
!   NOTE:  Any data that requires units (such as mass) is assumed to be in the consistent solver unit system.
!                 See Solving Units in the help system for more information.


kbc,0

```

Workshop 09.3 - Solution

- Add a Total Deformation result to the Solution branch
- Solve the model
  - The solve takes approximately 2 minutes.

Workshop 09.3 - Postprocessing

- Confirm the ramped loading by reviewing the Total Deformation result.



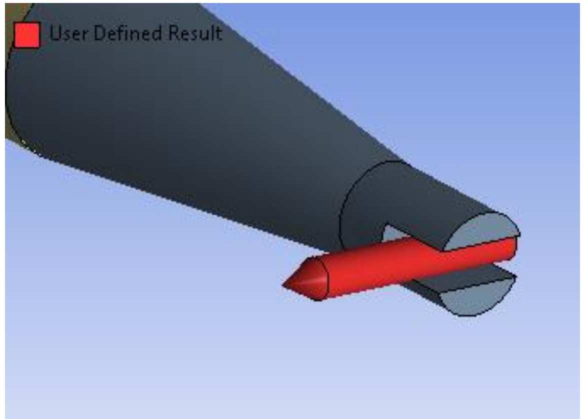
Note: your result magnitudes may vary slightly throughout this workshop due to mesh and software release differences

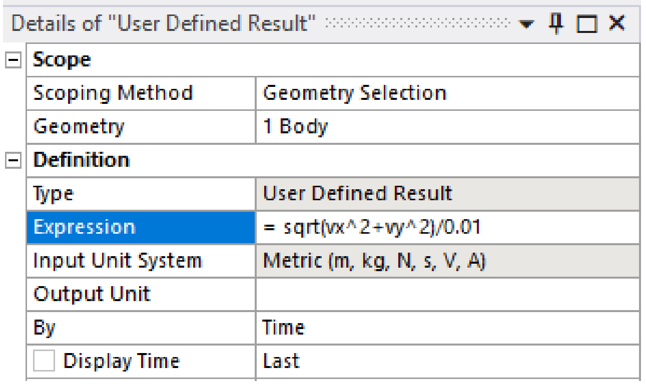
Workshop 09.3 - Postprocessing

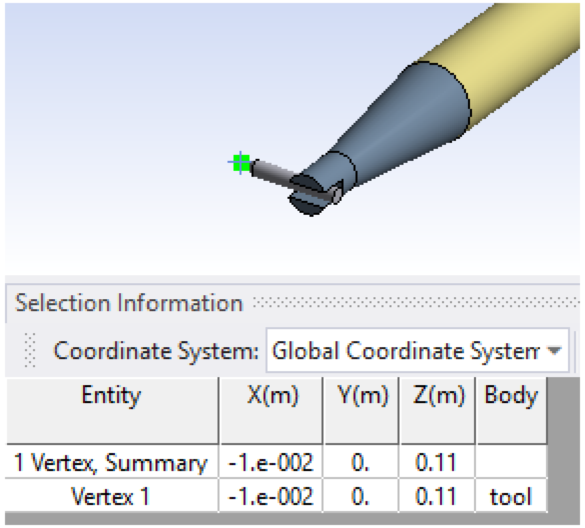
- Create a User Defined Result to confirm the applied rotational velocity, ω:
  - Use the vertex at the wire bonder tip as a reference point
  - Recall the following information:

$$|\mathrm{v}| = \omega . \mathrm{r}$$
$$|\mathrm{v}| = \left(\mathrm{v}_\mathrm{x}^2 + \mathrm{v}_\mathrm{y}^2\right)^{0.5}$$
$$\mathrm{r} = 0.01 \text{ m}$$

  - Create a user-defined result for $\omega = |\mathrm{v}|/\mathrm{r}$



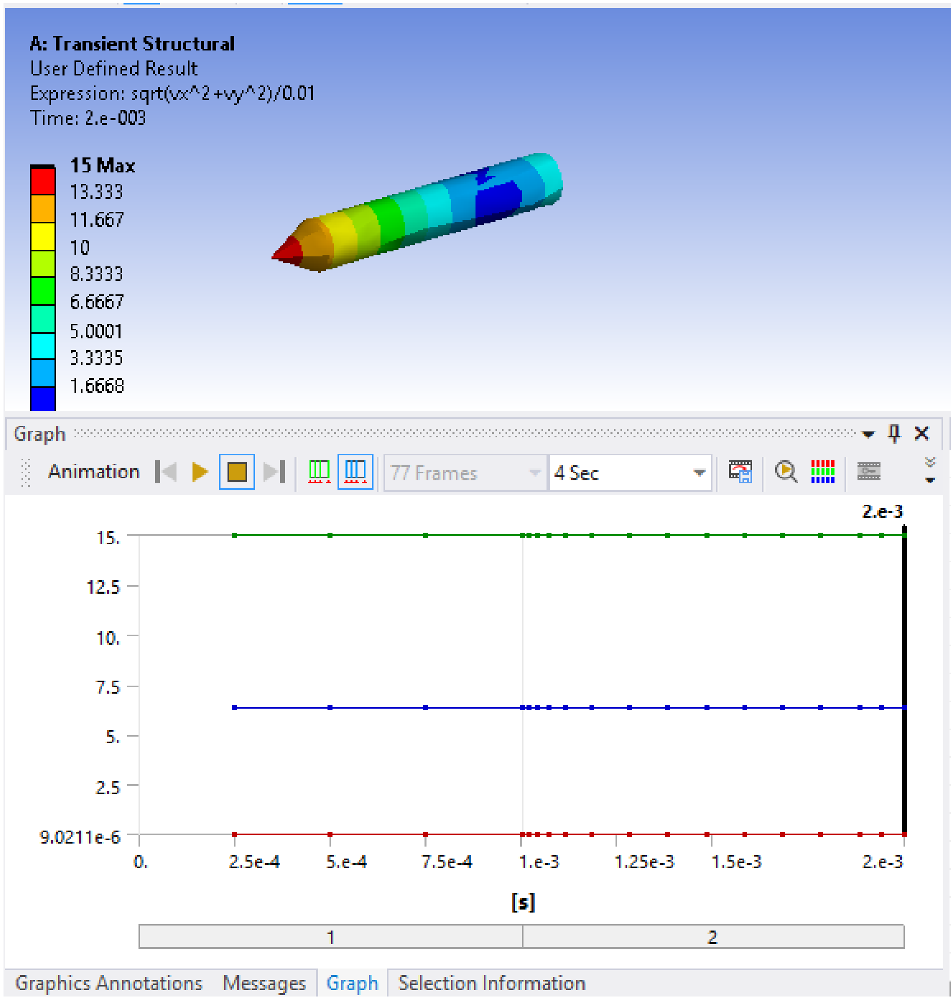




Ansys

Workshop 09.3 - Postprocessing

- Evaluate and review the computed values for ω:



	Time [s]	Minimum	Maximum	Average
1	2.5e-004	1.1401e-005	15.	6.3651
2	5.e-004	1.0794e-005	15.	6.3651
3	7.5e-004	9.0211e-006	15.	6.3651
4	1.e-003	1.013e-005	15.	6.3651
5	1.02e-003	2.8447e-003	15.	6.3652
6	1.04e-003	3.0033e-003	15.	6.365
7	1.07e-003	3.0714e-003	15.	6.3652
8	1.115e-003	2.1595e-003	15.	6.3651
9	1.1825e-003	2.2274e-003	15.	6.3651
10	1.2825e-003	1.106e-003	15.	6.3651
11	1.3825e-003	9.4506e-004	15.	6.3651
12	1.4825e-003	1.3836e-003	15.	6.3651
13	1.5825e-003	6.0203e-004	15.	6.3651
14	1.6825e-003	7.9109e-004	15.	6.3651
15	1.7825e-003	7.3832e-005	15.	6.3651
16	1.8825e-003	3.3856e-004	15.	6.3651
17	1.9413e-003	3.9896e-004	15.	6.3652
18	2.e-003	2.1725e-004	15.	6.3651

Click Here to see a demonstration of this workshop

Ansys

Ansys

End of presentation