Proceedings of the ASME-JSME-KSME Joint Fluids Engineering Conference 2015
AJK2015-FED
July 26-31, 2015, SEOUL, KOREA

AJK2015-02745

INVESTIGATION OF IMPELLER EXIT FLOW FIELD IN A HIGHLY LOADED CENTRIFUGAL COMPRESSOR

Soumya Dutta
Turbo Machinery and Engine Technology Department,
IHI Corporation, Yokohama, Japan

Wataru Sato
Turbo Machinery and Engine Technology Department,
IHI Corporation, Yokohama, Japan

Kenichi Funazaki
Department of Mechanical Engineering,
Iwate University, Morioka, Japan

Yukiya Kakudate
Department of Mechanical Engineering,
Iwate University, Morioka, Japan

ABSTRACT

Recently, the demand of a highly loaded impeller for industrial compressors is increasing due to environmental and economic reasons. In a compressor aerodynamic design including the matching of an impeller and a diffuser, it is very important to predict the impeller flow field precisely, but due to its high blade loading, a large flow separation and a strong clearance leakage flow occurring inside the impeller passage and impeller exit flow is distorted significantly. Thus it becomes difficult to predict a compressor performance with enough accuracy.

Therefore, in order to validate the prediction accuracy of CFD analysis to the highly loaded industrial compressor impeller, flow measurements at the impeller exit were conducted by using 3 holes yaw meter and hot wire anemometry traverse tests, and compared to the CFD results. CFD analysis was conducted by Iwate University using ANSYS CFX, in which SST model was applied to estimate turbulence viscosity.

From the comparison between CFD results and measurements, significant difference in turbulence intensity and flow angles between measurements and CFD results could be observed at the impeller exit. These results imply that the reason behind this difference could be because CFD prediction underestimates the free stream turbulence and mixing effects between free stream and clearance flows

NOMENCLATURE

b_3	Diffuser height
CFD	Computational Fluid Dynamics
D_2	Impeller Outer Diameter
Mu_2	Peripheral Mach number at impeller exit
P_s	Static Pressure
P_{tin}	Inlet Total Pressure
TKE	Turbulent Kinetic Energy
Tu	Turbulence Intensity
Z	Number of blades
ϕ	Flow coefficient

INTRODUCTION

Flow through a high pressure ratio centrifugal impeller has been a recurring challenge especially at high impeller rotational speeds as they are strongly affected by 3 dimensional boundary layers, secondary flows and flow separations [1]. In the context of modern day, the environmental and economic needs are driving the development of industrial compressors towards high efficiency and high pressure ratio. This requirement of high efficiency and pressure ratio, and the limitation in the maximum number of compressor stages gives rise to the necessity to increase a single stage centrifugal compressor impeller loading. Since diffusers are frequently applied to increase stage performance, an optimum diffuser design matched to a highly loaded impeller becomes essential.

However matching an impeller and a diffuser becomes a challenging task because the flow discharged by a highly loaded centrifugal impeller is highly distorted as extensively studied in [1, 2]. The unfavorable rotor exit flow is especially related to high flow angle differences from hub to shroud at impeller discharge which cause unsteady diffuser inlet flow conditions

[3, 4]. These effects originate from leading edge flow separations as well as strong secondary flows caused by tip clearance and shroud curvature. Thus in order to solve the problem of diffuser matching, impeller exit flow needs to be understood completely.

At high rotational speeds, high pressure rise in compressors produce supersonic velocities into the diffuser which in turn produces complicated flow with shock waves. In paper [5], hot wire measurements of impeller exit flow was undertaken, where impeller inducer is machined down so that the impeller retains the same diffusion, for measurement at a lower speed considering the vulnerability of hot wire measurements at transonic speeds. The low speed measurements of the outlet flow from the impeller were observed to be very similar to the flow at high speed. Similar to this, many other researches are dedicated to predicting impeller exit flow through the compressor at high rotational speed or simulating the flow conditions of high rotational speed at lower speeds by compressor redesign, however there are not so many researches focusing on impeller exit flow through the highly loaded compressor at a low operating speed.

In the present paper, instead of impeller redesign, the original impeller is rotated at a low rotational speed condition. In the current research, investigating the highly loaded impeller exit flow mechanism at a low rotational speed was considered as an essential first step to ultimately understand the high speed impeller exit flow mechanism.

Since CFD steady computations are extensively used in modern day, in the field of centrifugal compressor design, it is of utmost necessity to identify the range of usability and limitations of using CFD as a tool for product design and development process. The purpose of the current study is improvement of our insight into the real flow mechanism at impeller exit in case of a highly loaded centrifugal impeller. This leads to the objective of current research to primarily clarify the similarities and differences in impeller exit flow characteristics, through comparison of simulation (CFD) and test results and understand the reason that can cause such difference in prediction.

COMPRESSOR CONFIGURATION

The compressor impeller chosen for the current study is shown in Tab. 1 and Fig. 1. The diffuser used in the current configuration is vane less.

TABLE 1. MAIN DIMENSIONS OF COMPRESSOR

Z	8/8, full/splitter
D_2	250 mm
b_3	10.65 mm

FIGURE 1. IMPELLER

EXPERIMENTAL METHOD

To understand the flow mechanism exiting a highly loaded impeller, measurements on flow field at impeller exit flow was undertaken as the primary task.

In the current research, Yaw meter pressure probes and hot wire anemometer probes were applied to measure the absolute flow angle, total pressure and turbulence intensity at several circumferential locations at the impeller exit streamwise position of 1.1 times the impeller diameter (Fig. 2).

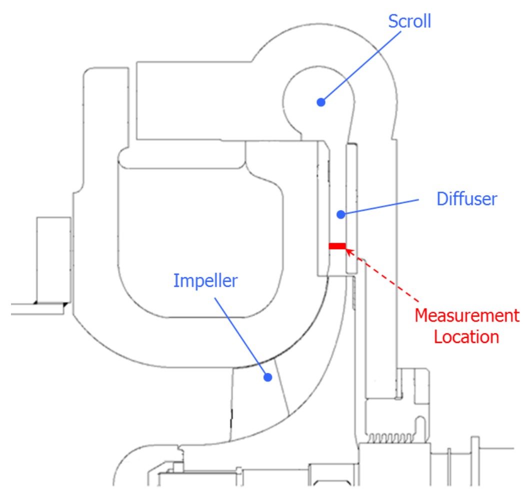


FIGURE 2. FLOW MEASUREMENT LOCATION 1.1 D2 CIRCUMFERENTIAL POSITION

The measurement of impeller exit flow field parameters at various span-wise locations, at a constant 1.1D2 stream-wise location shown as the measurement location in Fig. 2, could be achieved by means of traversing the probes in the span-wise direction using a traversing device at a precision of 0.01 mm . Triggering of the signal at an individual blade passing event is also applied in the measurement which is done using a gap sensor.

Measurement of flow angles and total pressure

A 3-hole yaw meter probe is used for measuring absolute flow angles and total pressure. Yaw meter probes measure the flow angle from the yaw angles of the probe. Alignment to the flow direction is detected as a yaw angle value of 0^{o}, which occurs when the difference of the total pressure reading of the 2 holes on each sides of the center hole is equal to 0. Extensive yaw meter measurements were done by repeatedly rotating the yaw meter by 0.1^{o}until the total pressure difference value of 0 is reached. This repetitive measurement process of finding the exact impeller exit flow angle was undertaken at each measurement locations for all rotational speeds. Thus using this process yaw meter could detect all possible flow angles at the impeller exit, even a reverse flow angle. The span wise measurement locations at the stream wise position of 1.1 D2 are shown in the following Fig. 3.

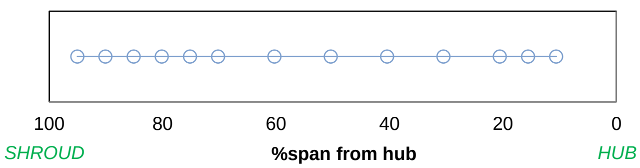


FIGURE 3. SPAN WISE MEASUREMENT LOCATIONS

Yaw meter measurements with throttle ring were done at the measurement location shown in Fig. 3 using the current highly loaded impeller at Mu2 = 0.67 and Mu2 = 1.18 at three flow coefficients individually.

Measurement of Turbulence

Turbulence intensity measurement and its comparison with the turbulence predicted by the turbulence model applied in case of the CFD computation, is an effective way to understand the differences in impeller exit flow characteristics predicted by simulations (CFD) and measurements. Hot wire anemometry for turbulence intensity prediction is generally used in case of low velocity flows for point data measurements [6]. Sensitivity to heat or high velocity flows renders hot wire anemometry limited to a certain range of measurements. [7]

In current research, hot wire probes that are used for flow measurements are X probes which consists of 2 wires perpendicular to each other and are capable of simultaneous measurements. The measurement position is similar to the yaw meter at 1.1 times D2 circumferential position. The X probe traversed through 11 points from shroud to hub in order to record the span-wise variation flow field values. Hot wire measurements with throttle ring were done at the measurement locations shown in Fig. 4 at Mu2 = 0.67 and Mu2 = 0.88 at three flow coefficients individually.

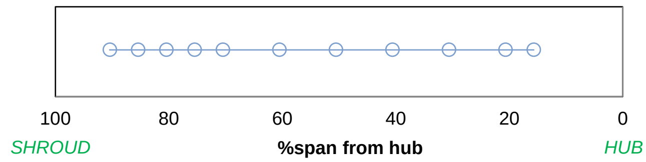


FIGURE 4. SPAN WISE MEASUREMENT LOCATIONS

Turbulence Intensity is derived from the fluctuation in velocities recorded by the hot wire over a period of time. Since the hot wire probes record the two dimensional flow velocities, turbulent kinetic energy is calculated by the following equations.

$$TKE = \frac{1}{2}(\tilde{u}^{2} + \tilde{v}^{2})$$

$$\tilde{u}^{2} = \frac{1}{N}\sum(u-\bar{u})^{2}, \tilde{v}^{2} = \frac{1}{N}\sum(v-\bar{v})^{2} \tag{1}$$

Here u, v are the Impeller phase locked instantaneous velocities; $\bar{u}, \bar{v}$ are the Impeller phase locked ensemble averaged velocities and N signifies the total number of readings. $\tilde{u}^{2}, \tilde{v}^{2}$ are known as variance which is a measure of the dispersion of data about a mean value. Turbulence Intensity (Tu) is a measure of turbulence level calculated by the following Eq. (2) where $\bar{U}$ is the area average of time averaged absolute velocity at each measurement points and TKE is the turbulent kinetic energy calculated by Eq. (1).

$$Tu = \frac{\sqrt{TKE}}{\bar{U}} \times 100 \tag{2}$$

Application of Throttle ring

Validation of experimental results with that of CFD becomes difficult especially regarding the flow at impeller exit, considering that CFD computations are single passage computations assuming axis symmetric flow and don't include the scroll in computational domain. This is the reason why, a throttle ring was applied to the experimental setup at the diffuser exit as shown in fig 5.

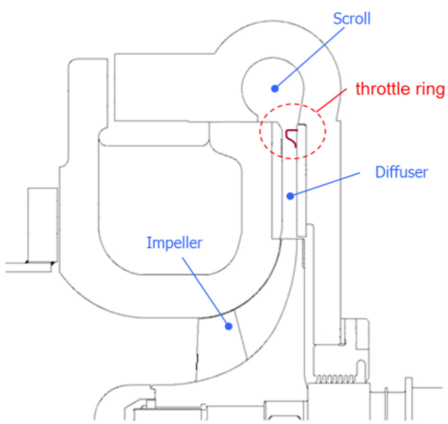


FIGURE 5. MERIDIONAL VIEW OF THE EXPERIMENTAL SETUP

The throttle ring was used for increasing the axis symmetric flow condition and decreasing the effect of scroll on flow field at impeller exit.

Data Processing

Data recorded in the yaw meter and hot wire anemometry measurements are analyzed for validation using MS Excel and MATLAB programming. In case of hot wire anemometry, the data samples obtained by point measurements at various span and circumferential positions were analyzed using phase locking and ensemble averaging techniques. The signal of a gap sensor acted as the trigger recording every blade passing event. The hot wire measurement data is triggered into samples of data containing the circumferential component of flow velocities at a specific span position, which is further measured for all span positions. Ensemble averaging flow velocity values at every point result in the turbulence intensity spanning the 2D flow field at impeller exit.

The initial turbulence intensity data recorded showed a small contrast between the regions of high and low turbulence. This was due to the fact that, the number of samples obtained by phase locking and later used for ensemble averaging were low and hence the authenticity of the flow mechanism predicted by the results was insufficient. Therefore the number of samples was increased for data recorded per sample. Increase in the number of samples helped improve the contrast between the regions of high turbulence compared to the regions of low turbulence.

COMPUTATIONAL ANALYSIS

The computational analysis on the current compressor was performed using ANSYS CFX steady simulations at Iwate University. Single passage RANS flow computations were performed under the axis symmetric flow assumption. SST Turbulence model was chosen for the steady computation.

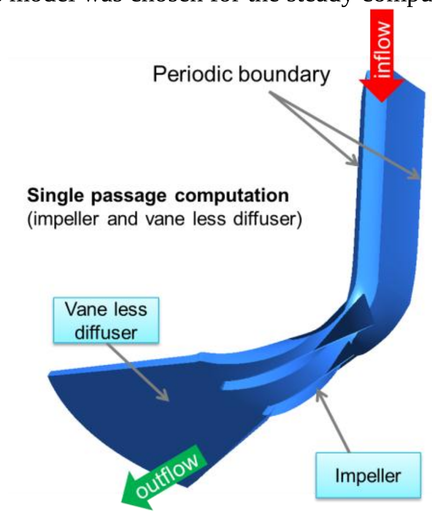


FIGURE 6. CFD DOMAIN

The total number of mesh was 4 million for the centrifugal impeller and vane less diffuser setup. The boundary conditions used are given in the following Tab. 1.

TABLE 2. BOUNDARY CONDITIONS

Inlet total Pressure	101325 Pa
Inlet Total Temperature	288.15 K
Outlet flow coefficient	0.048

EXPERIMENTAL RESULTS AND VALIDATION

The effect of throttle ring in creating an axis symmetric flow field at the impeller exit could be realized (Fig. 7).

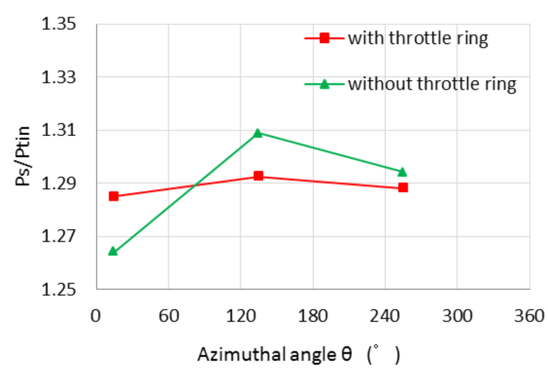
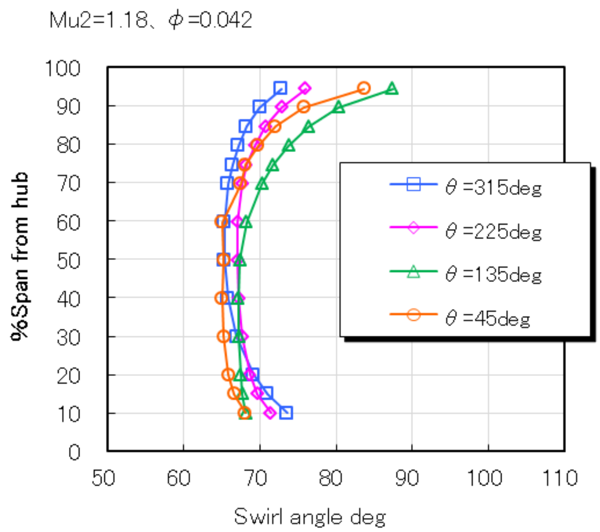


FIGURE 7. CIRCUMFERENTIAL VARIATION OF STATIC PRESSURE

It was observed that the effects of throttle ring in creating an axis symmetric flow field at the impeller exit could be realized to a greater extent in case of the throttle ring choke flow point rather than the lower flow coefficients (Fig. 8).



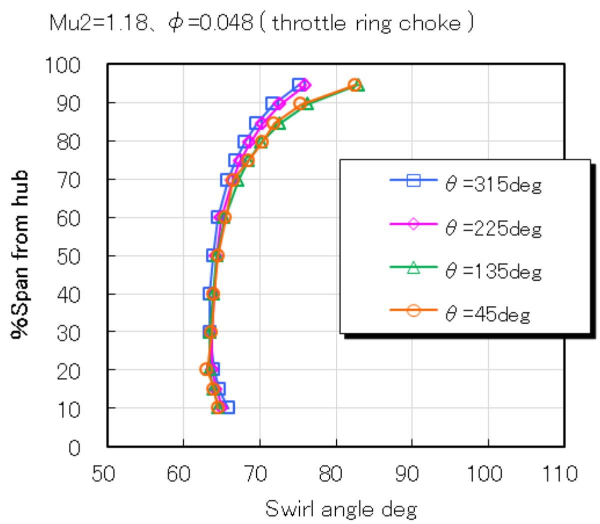
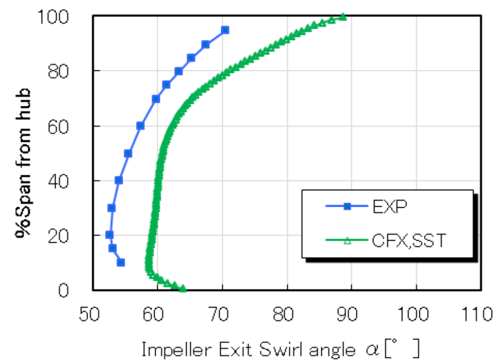


FIGURE 8. IMPELLER EXIT FLOW ANGLE CIRCUMFERENTIAL VARIATION (θ = AZIMUTHAL ANGLE)

The results of yaw meter measurements show flow angle increasing from hub to shroud at impeller outlet. This trend is also seen in CFD results however, significant decrease in impeller exit flow angles compared to the CFD results. It can be observed in Fig. 9 that CFD results overestimate the absolute flow angle values throughout all span positions at Mu2=0.67, especially at the shroud region. The absolute pressure ratio shows similar trend in measurement results as in CFD and the values are also similar to each other.



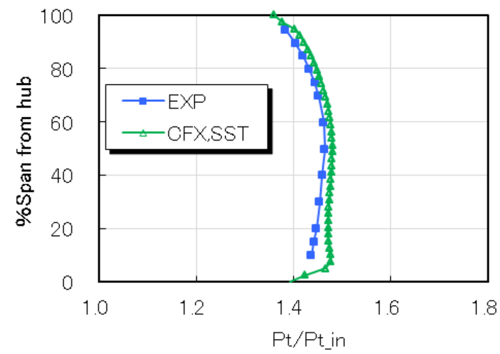


FIGURE 9. COMPARISON AT MU2 = 0.67, Φ = 0.048

At higher rotational speeds (at Mu2 =1.18) overestimation of absolute flow angle by CFD seems to increase towards the shroud (Fig. 10). CFD also predicts a reverse flow at the shroud, which is not detected in case of experiments. Pressure ratio comparison show similar trend in CFD against the experimental results like in the case of lower rotational speed, however the relative deviation of CFD result from experiment seems higher in this case.

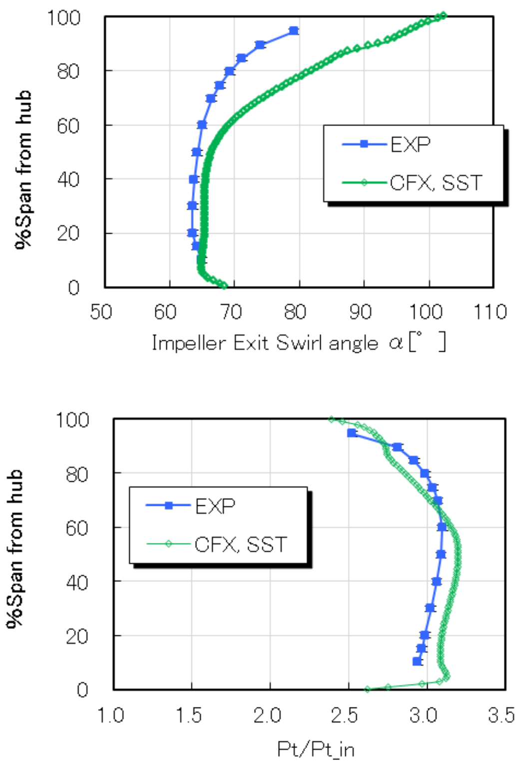


FIGURE 10. COMPARISON AT MU2 = 1.18, Φ = 0.048.

The results of hot wire anemometry at the lower rotational speed, (Mu2 = 0.67) show resemblance with CFD results near the shroud region (Fig. 11), region of high turbulence can be observed at the shroud in both measurements and CFD results.

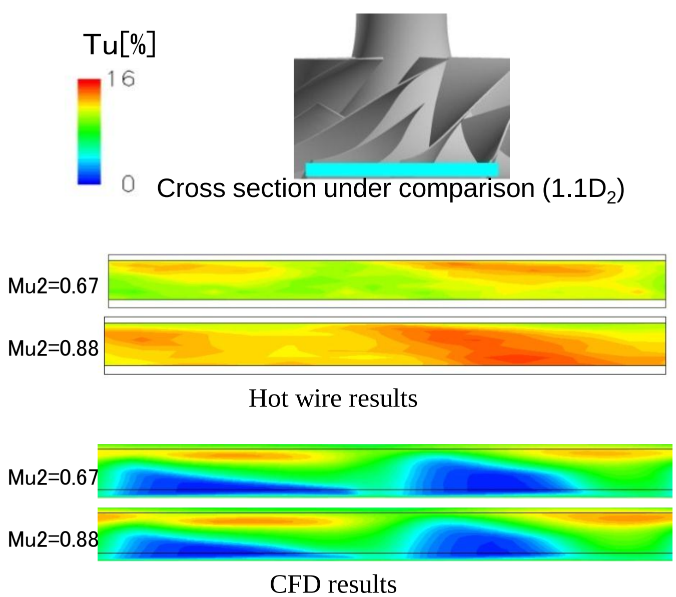


FIGURE 11. HOTWIRE RESULTS

However CFD results predict regions of lower turbulence at the hub whereas no such regions exist in the measurement results. This could be due to the fact that CFD results underestimate the free stream turbulence and mixing effects between free stream and clearance flows exist. Significant increase in turbulence intensity can be observed in measurement results at a higher rotational speed (Mu2 = 0.88).

CONCLUSIONS

A trend of growing difference in the impeller exit flow angle values from hub to shroud is observed in the Yaw meter measurement results when compared to CFD results. CFD results predict the presence of a strong reverse flow at the shroud region, differing from measurement results for high speed impellers.

When turbulent intensity measured using hot wire anemometry at the impeller exit position 1.1 times D2 at all span positions is compared to CFD results, resemblance could be observed in the shroud region where a region of high turbulence exists. However CFD results predict a region of lower turbulence at the hub whereas in measurements, no such region exists. This could be due to the fact that CFD results underestimate the free stream turbulence and mixing effects between free stream and clearance flows.

REFERENCES

[1] Ekardt, D.," Instantaneous Measurements in the Jet and Wake Discharge Flow of a Centrifugal Compressor Impeller," ASME JOURNAL OF ENGINEERING FOR POWER, Vol.97, 1975, pp. 337-346.

[2] Senoo,Y., and Ishida,M.,"Behaviour of Severly Asymmetric Flow in a Valeless Diffuser." ASME JOURNAL OF ENGINEERING FOR POWER, Vol.97, 1975, pp.375-387.

[3] Eisenlohr, G., Dalbert, P., Krain, H., Pröll, H., Richter,F.A., Rohne, K.H., Analysis of the Transonic Flow at the Inletof a High Pressure Ratio Centrifugal Impeller, ASMEConference, Stockholm, Sweden, June, 1998, ASME-Paper98-GT-24, p.11.

[4] Eisenlohr, G., Krain, H., Richter, F.A., Tiede, V.„Investigations of the Flow Through a High Pressure RatioCentrifugal Impeller". ASME-Paper GT-2002-30394, p. 9.

[5] Inoue,M., and Cumptsy, N.A.,1984,"Experimental Study of the Centrifugal Impeller Discharge Flow in Vaneless Diffusers," ASME Journal of Engineering for Gas Turbines and Power, Vol. 106, pp. 455-467.

[6] Pinarbasi, A., and Johnson, M.W., 1994a "Detailed Flow Measurements in a Centrifugal compressor Vaneless Diffuser" ASME Journal of Turbomachinery. v:116, no.3, pp.453-461.

[7] Tech-report, A., 2003. "HOW TO MEASURE TURBULENCE WITH HOT-WIRE ANEMOMETERS" Dantec Dynamics A.